Altair OptiStruct®

Concept Design with Topology and Topography Optimization

Altair Engineering

April, 2009

Day 1 Agenda

- **Introduction**
- **Theoretical Background**
- **Optimization Interface and Setup**
- **Concept Design**
  - **Topology Optimization**
    - Exercise 4.1: Topology Optimization of a Hook with Stress Constraints
    - Exercise 4.2: Topologic Optimization of a Control Arm
  - **Topography Optimization**
    - Exercise 4.3: Topography Optimization of a Slider Suspension
  - **Free-size Optimization**
    - Exercise 4.4: Free-size Optimization of Finite Plate with Hole

Day 2 Agenda

- **Review**
- **Fine Tuning Design**
  - **Size Optimization**
    - Exercise 5.1 – Size Optimization of a Rail Joint
  - **Shape Optimization**
    - Exercise 5.2: Shape Optimization of a Rail Joint
  - **Free-shape Optimization**
    - Exercise 5.3 - Free-shape optimization Compressor Bracket

Chapter 1 - Introduction

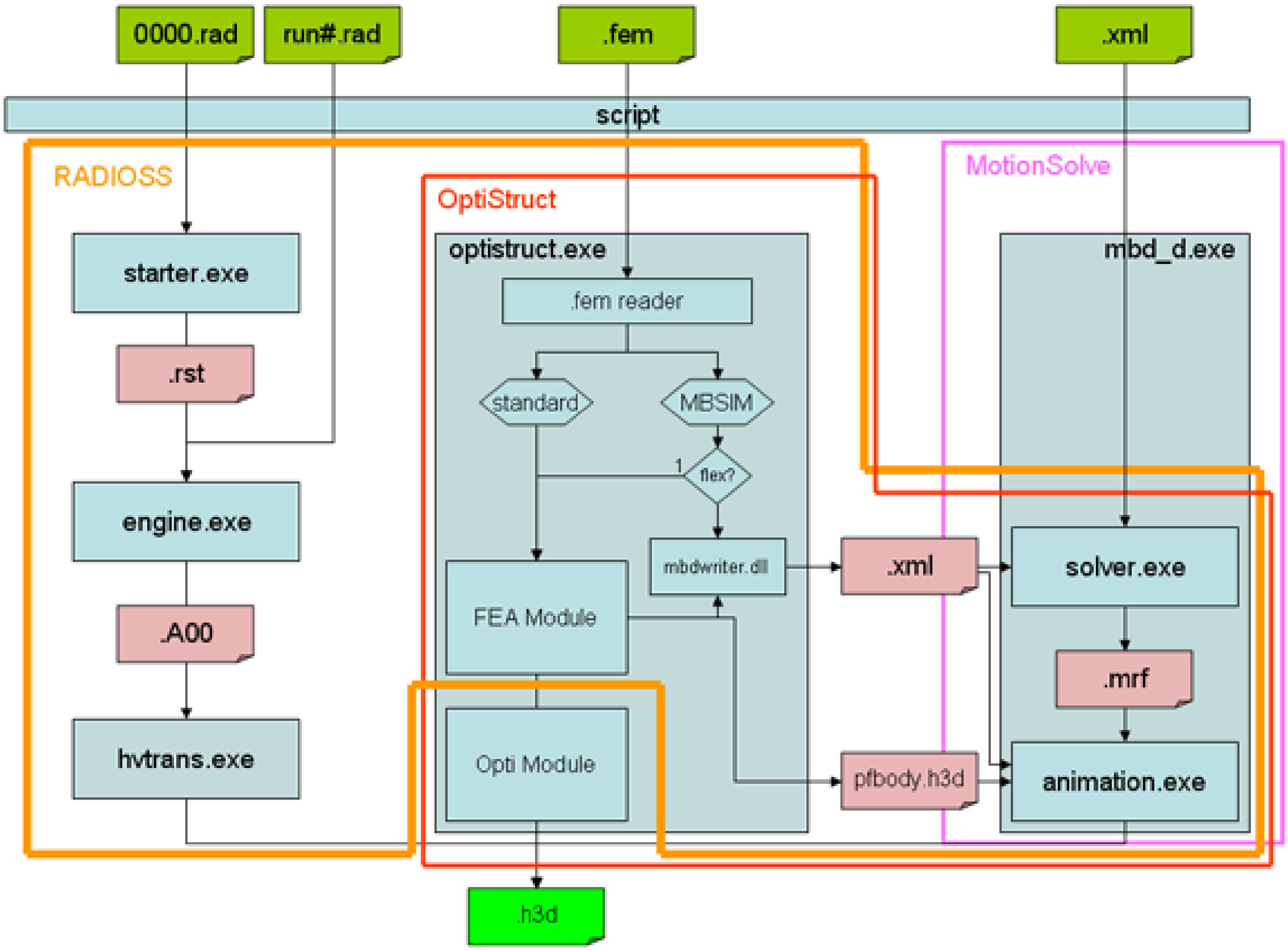
HyperWorks Overview

OptiStruct Overview

HyperWorks Overview

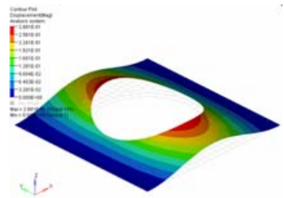
- Modeling
- Analysis
- Optimization
- Visualization
- Reporting
- Performance data management.

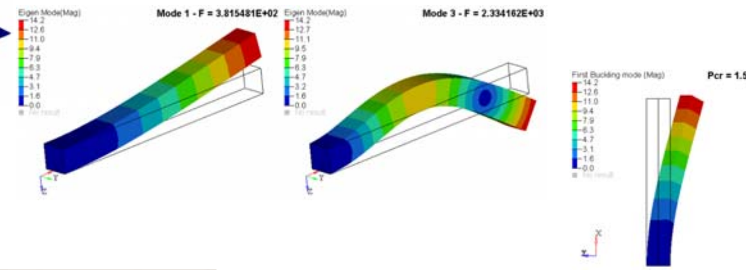
OptiStruct in HyperWorks



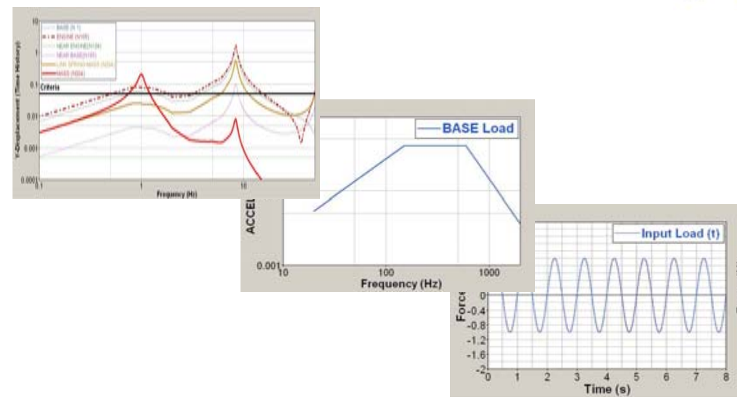
OptiStruct Overview

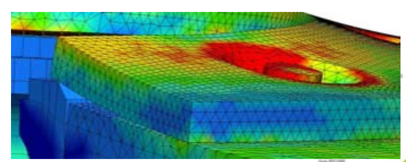
Finite Elements Analysis

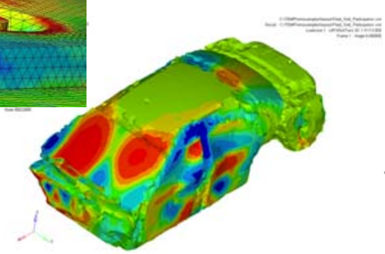


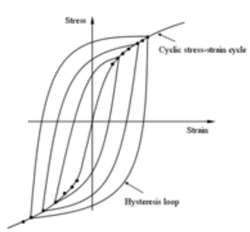


- **Basic analysis features**
  - Linear static analysis.
  - Normal modes analysis.
  - Linear buckling analysis.
  - Thermal-stress steady state analysis
- **Advanced analysis features**
  - Frequency response function (FRF) analysis
    - Direct
    - Modal
  - Random response analysis
  - Transient response analysis
    - Direct
    - Modal
  - Transient response analysis based on the Fourier method
    - Direct
    - Modal
  - Non-linear contact analysis
  - Acoustic Analysis (Structure and Fluid)
  - Fatigue Analysis (σN and εN)



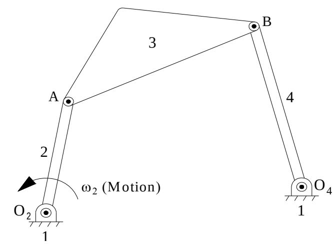




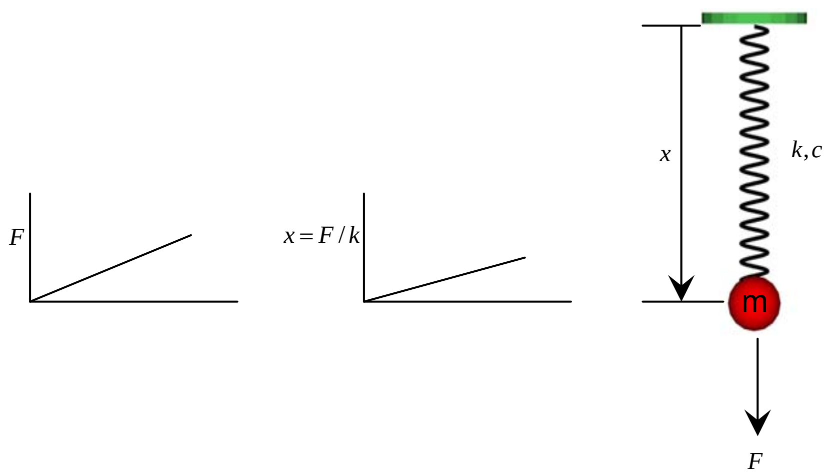


OptiStruct Overview

MBD Analysis



- Kinematics
- Static
- Quasi-static
- Dynamics



OptiStruct Optimization Overview

Design Process

Topology

Free-size

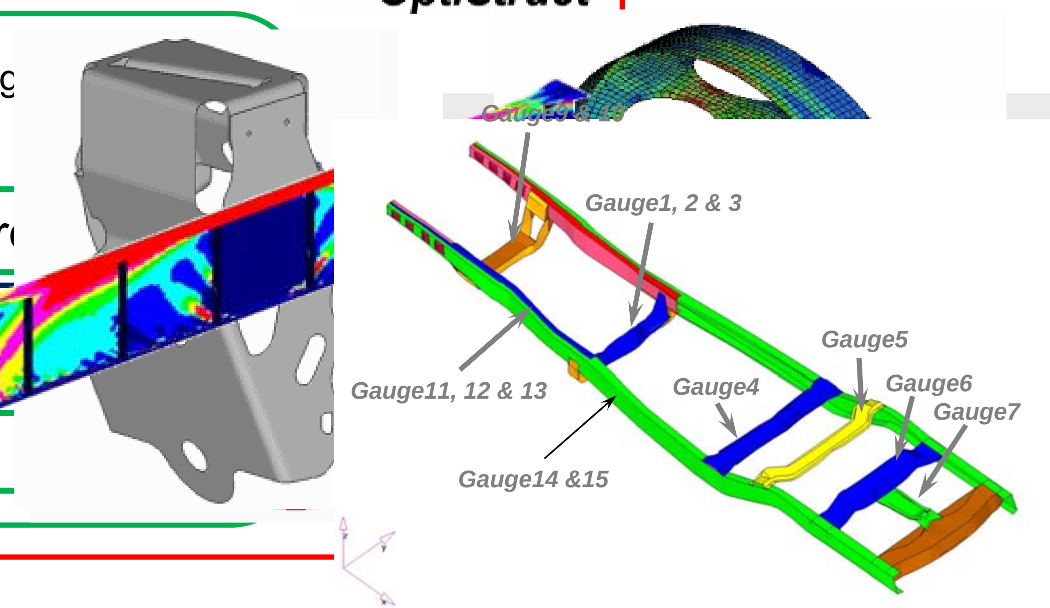
Topography

Concept Level Design

Integrated FEA Solver

Desig

Fr



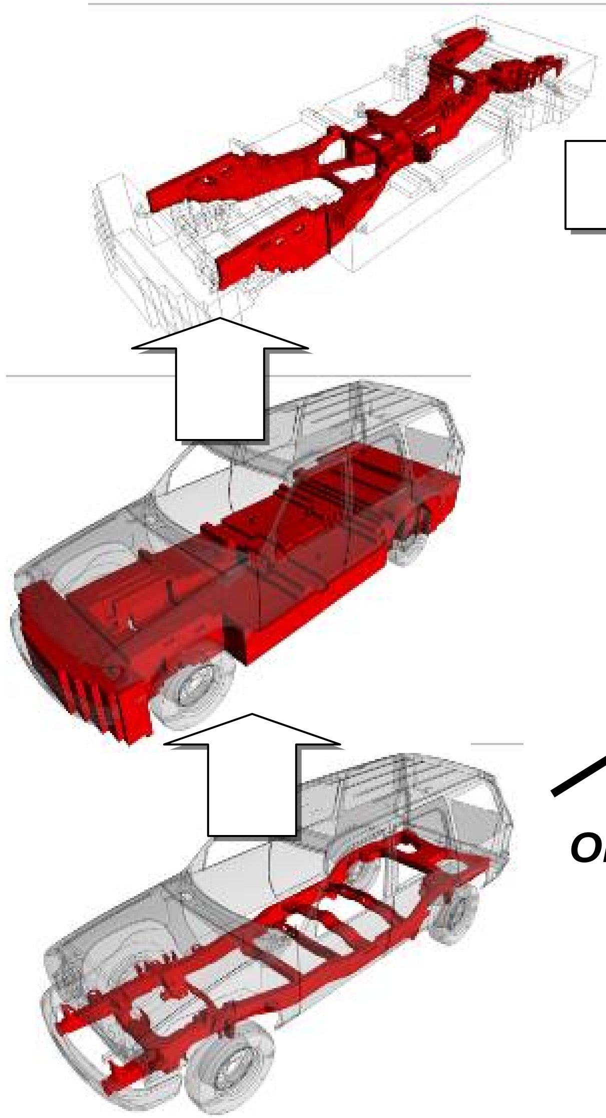
Solver Neutral

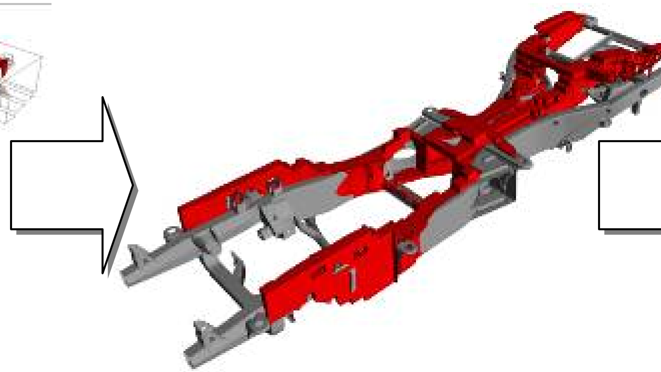
DOE

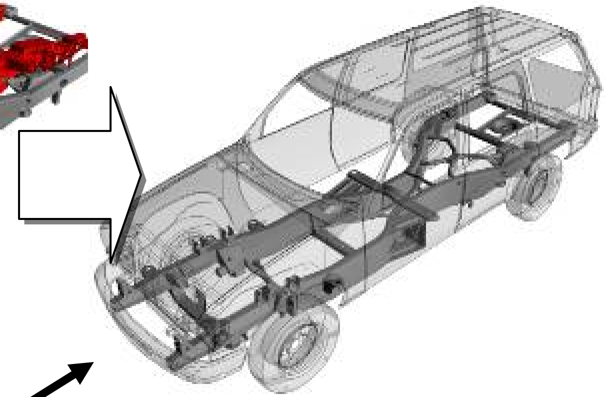
Approximations

Stochastic Studies

Lightweight SUV Frame Development







New Frame

Old Frame

Mass reduction: 20%

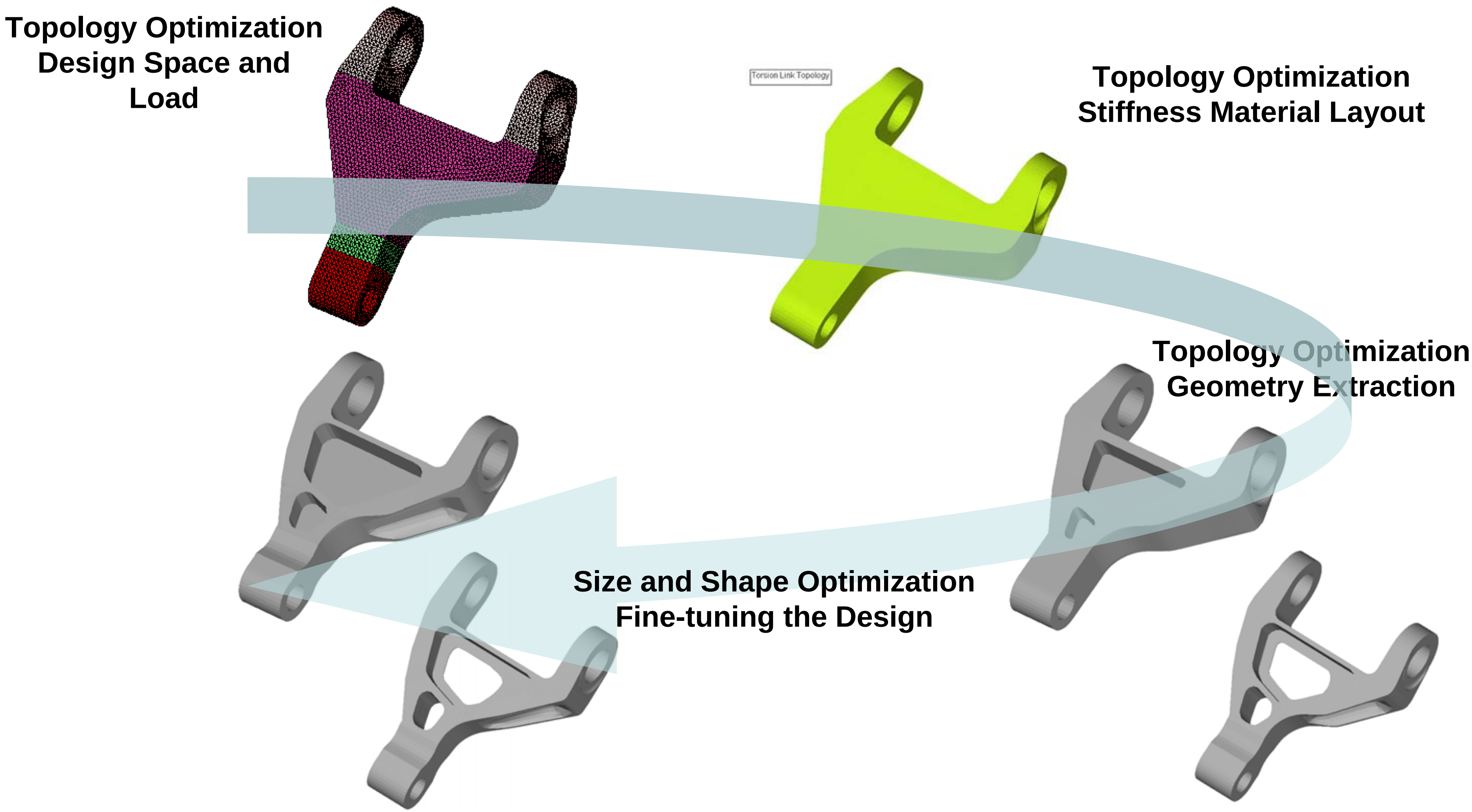
Increase torsion stiffness: 31%

Weld length reduction: 50%

The Auto/Steel Partnership
Member Companies

Bethlehem Steel Corporation
Dofasco Inc.
General Motors Corporation
National Steel Corporation
Stelco Inc.
Daimler Chrysler Corporation
Ford Motor Company
Ispat Inland Inc.
Rouge Steel Corporation
United States Steel Corporation

Optimization Process for Torsion Links



Upper and lower link mass without pins is down to 176 lbs from 240 lbs.

Chapter 2 – Theoretical Background

Optimization

Optimization Concepts and Definitions

Structural Optimization Concepts

The Optimization Problem Statement:

- Objective *(What do I want?)*

  min f(x) also min [max f(x)]

- Design Variables *(What can I change?)*

  $X_i^L \leq X_i \leq X_i^U \quad i = 1,2,3,\ldots N$

- Design Constraints *(What performance targets must be met?)*

  $g_j(x) \leq 0 \qquad j = 1, 2, 3, \ldots, M$

Note: The functions f(x), $g_i(x)$, can be linear, non-linear, implicit or explicit, and are continuous

Example: Explicit $y(x) = x^2 - 2x$

Implicit $y^3 - y^2x + yx - \sqrt{x} = 0$

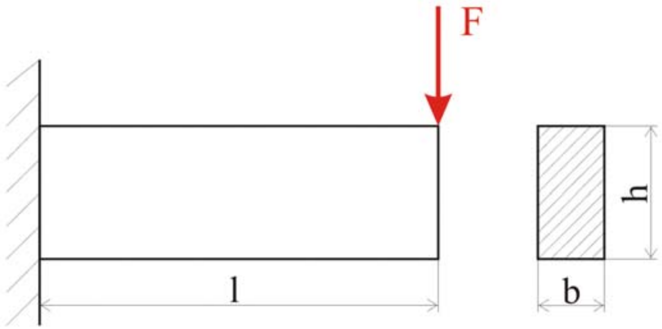
Optimization Definitions

- ***Topology:*** is a mathematical technique that optimized the material distribution for a structure within a given package space
- ***Topography:*** Topography optimization is an advanced form of shape optimization in which a design region for a given part is defined and a pattern of shape variable-based reinforcements within that region is generated using OptiStruct .
- ***Free Size:*** is a mathematical technique that produces an optimized thickness distribution per element for a 2D structure.

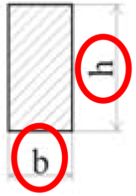
Optimization Definitions

- **Shape:** *is an automated way to modify the structure shape based on predefined shape variables to find the* optimal shape.
- **Size:** *is an automated way to modify the structure parameters (Thickness, 1D properties, material properties, etc…) to find the* optimal design.
- **Gauge:** *is a particular case of size, where the DV are 2D props (Pshell or Pcomp)*
- **Free Shape:** *is an automated way to modify the structure shape based on set of nodes that can move totally free on the boundary to find the* optimal shape.
- **Composite shuffle:** *is an automated way to determine the optimum laminate stack sequence. DVs are the plies sequence of stacking. It is used for composite material only defined using PCOMP(G) or PCOMPP.*

Optimization Terminology



- ***Design Variables:*** System parameters that are varied to optimize system performance.
- ***Design Space:*** selected parts which are designable during optimization process. For example, material in the design space of a topology optimization.



20 < b < 40

30 < h < 90

Optimization Terminology

Response: Measurement of system performance. σ *(b,h)* ; τ*(b,h), mass*

- DRESP1
  - Simple response definition
  - Mass, mass fraction, volume, volume fraction, compliance, frequency, displacement, stress, strain, force, composite responses, weighted compliance, weighted frequency, and compliance index, frequency response analysis responses
- DRESP2
  - Response definition using a user defined function
  - Defines responses as function of design variables, grid location, table entries, responses, and generic properties

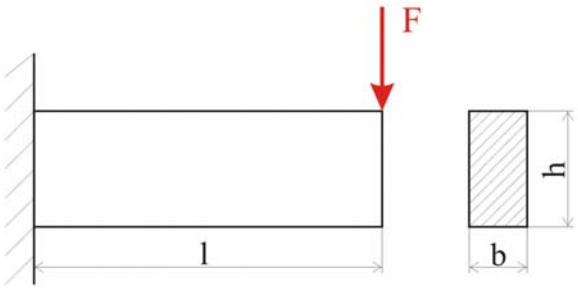
Example: Average displacement of two nodes:

$$F(x_1, x_2) = \frac{x_1 + x_2}{2}$$

Where x1, x2 are nodal displacements

- DRESP3
  - Response definition using a user defined external function
  - External function may be written in C (C++) or Fortran

Optimization Terminology



- ***Objective Function:*** Any response function of the system to be optimized. The response is a function of the design variables. Ex. Mass, Stress, Displacement, Moment of Inertia, Frequency, Center of Gravity, Buckling factor, and etc.

$$\mathit{min}\ \textbf{Weight}(b,h)$$

- ***Constraint Functions:*** Bounds on response functions of the system that need to be satisfied for the design to be acceptable.

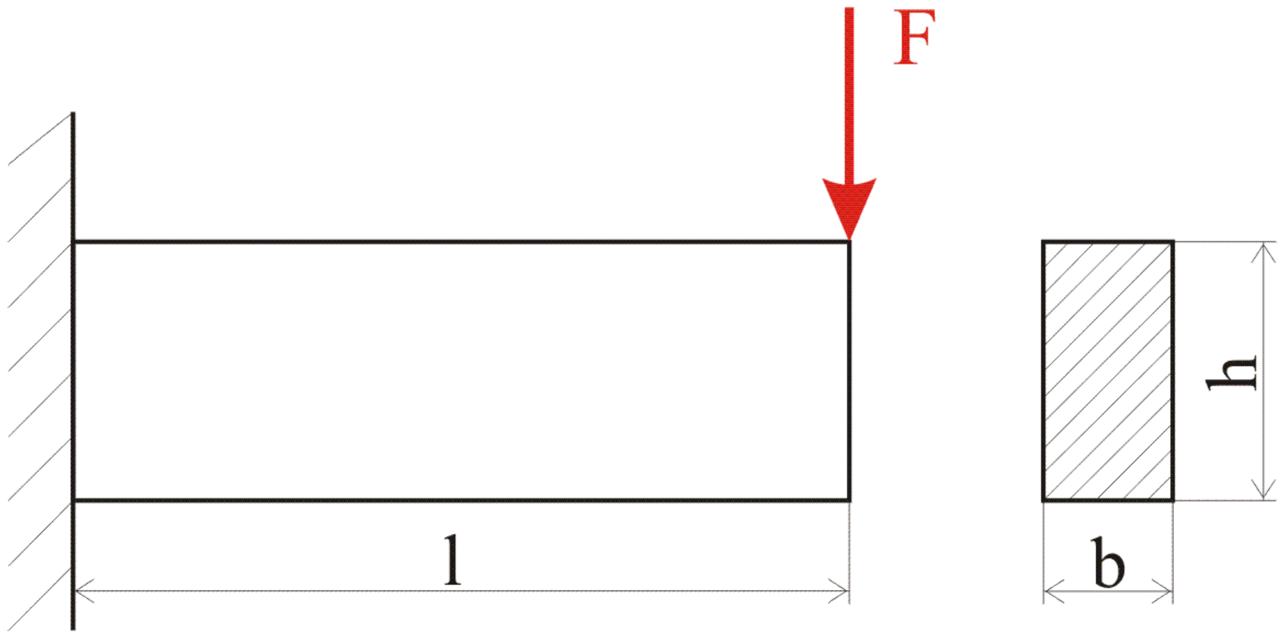
$$\sigma(b,h) \leq 70\ \text{MPa}$$

$$\tau(b,h) \leq 15\ \text{MPa}$$

$$h \geq 2*b$$

Optimization Problem Example

- A cantilever beam is modeled with 1D beam elements and loaded with force F=2400 N. Width and height of cross-section are optimized to minimize weight such that stresses do not exceed yield. Further the height h should not be larger than twice the width b.



Optimization Problem Example

- Objective
  - Weight: *min m(b,h)*
- Design Variables
  - Width: $b^L < b < b^U$, $20 < b < 40$
  - Height: $h^L < h < h^U$, $30 < h < 90$
- Design Region: *All beam elements*
- Design Constraints:

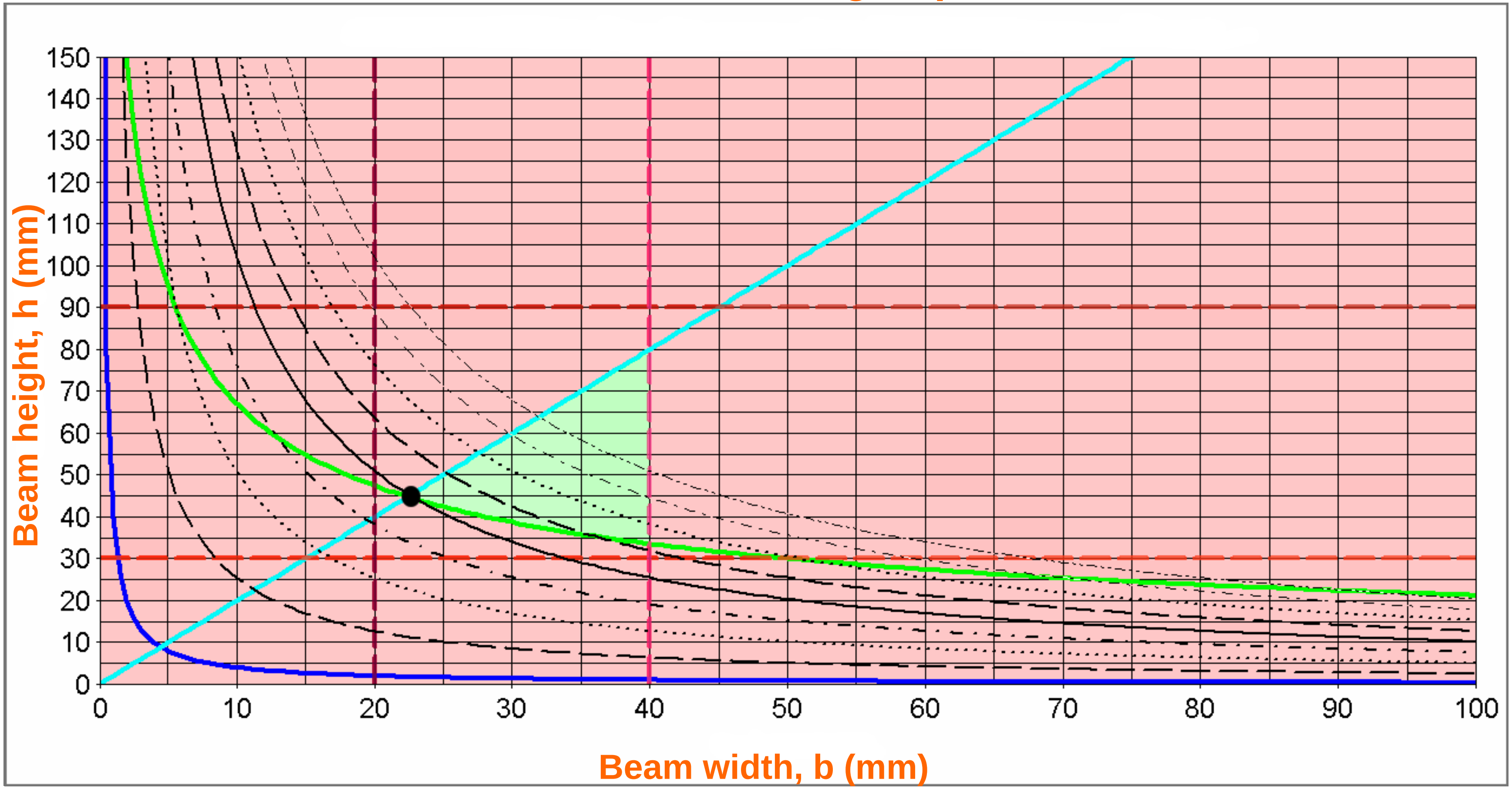
$$\sigma(b,h) \leq \sigma_{max}, \text{ with } \sigma_{max} = 160 \text{ MPa}$$

$$\tau(b,h) \leq \tau_{max}, \text{ with } \tau_{max} = 60 \text{ MPa}$$

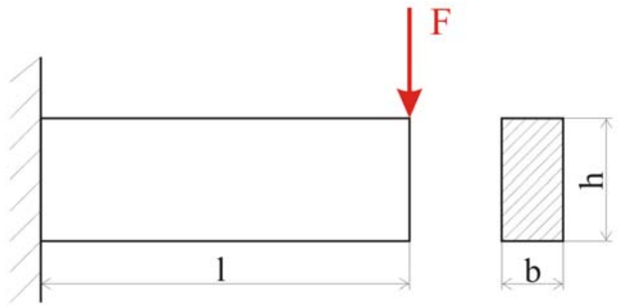
$$h \leq 2*b$$

Optimization Problem Example

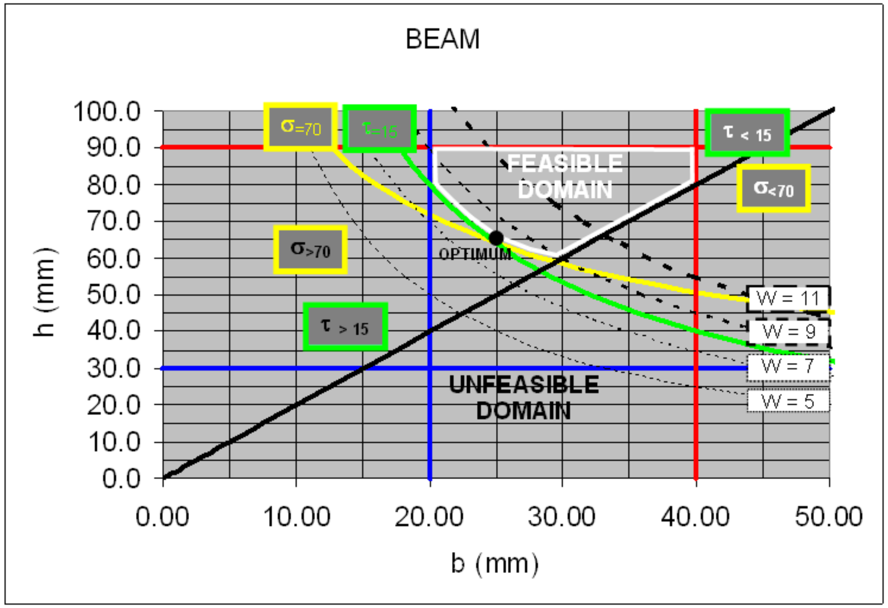
Mathematical Design Space



Optimization Terminology



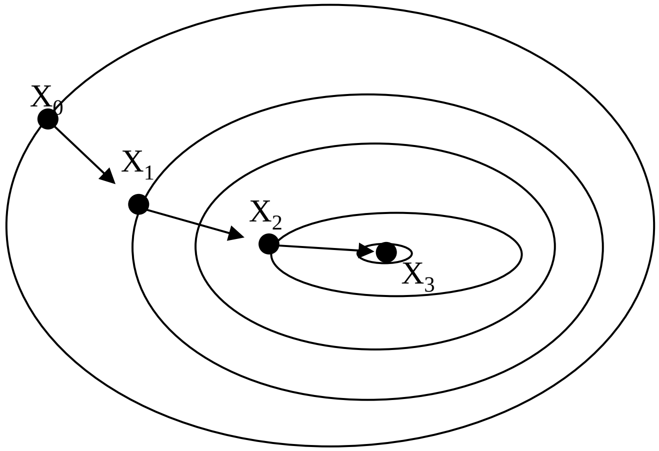
- ***Feasible Design:*** One that satisfies all the constraints.
- ***Infeasible Design:*** One that violates one or more constraint functions.
- ***Optimum Design:*** Set of design variables along with the minimized (or maximized) objective function and satisfy all the constraints.



Cantilever beam problem (Optimum (b=24.9, h=64.3), W = 8).

Gradient-based Optimization

1. Start from a X0 point
2. Evaluate the function F(Xi) and the gradient of the function ∇F(Xi) at the Xi.
3. Determine the next point using the negative gradient direction: $X_{i+1} = X_i - \gamma \nabla F(X_i)$.
4. Repeat the step 2 to 3 until the function converged to the minimum.



Optimization Terminology

Sensitivity Analysis

- ***Direct*** $\mathbf{K}\frac{\partial \mathbf{u}}{\partial x} = \frac{\partial \mathbf{f}}{\partial x} - \frac{\partial \mathbf{K}}{\partial x}\mathbf{u}$
  - Low number of Dvs
  - High number of constraint
- ***Adjoint*** $\frac{\partial g}{\partial x} = \frac{\partial \mathbf{q}^T}{\partial x}\mathbf{u} + \mathbf{a}^T\left[\frac{\partial \mathbf{f}}{\partial x} - \frac{\partial \mathbf{K}}{\partial x}\mathbf{u}\right]$
  - High number of DVs
  - Low number of constraint

Size and shape

Topology

Move Limit Adjustments $\underline{\mathbf{x}} \leq \underline{\mathbf{x}}_m \leq \mathbf{x} \leq \overline{\mathbf{x}}_m \leq \overline{\mathbf{x}}$

Constraint Screening

Regions and Their Purpose

Discrete Design Variables

Interpreting the Results

- Objective
  - Did we reach our objective?
  - How much did the objective improve?
- Design Variables
  - Values of variables for the improved design
- Constraints
  - Did we violate any constraints?

Interpreting the results

What can go wrong?

- Local minimum vs. global minimum
- Solution might not be available with the given objective, constraints and design variables – over constrained
- Efficiency of Optimization
  - Relation between constraints and design variables wrt their numbers
- Unconstrained Optimization Problem
  - Optimization problem setup is not appropriate
- Issues related to FEA modeling
  - Stress constraints on nodes connected to rigids

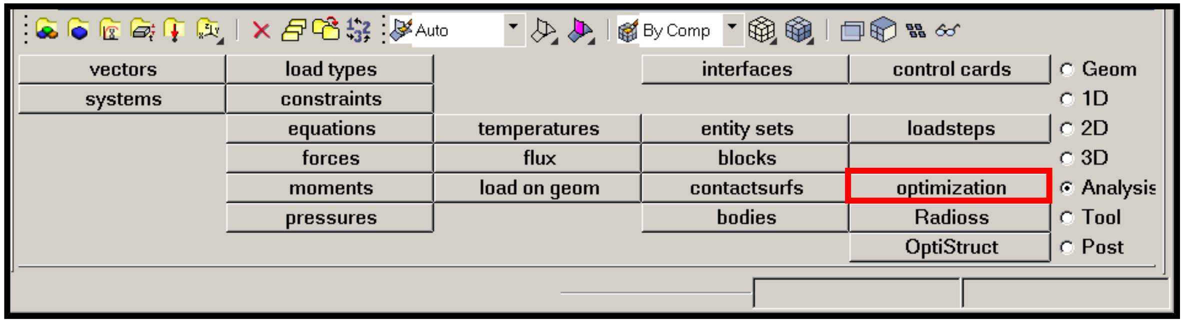
Chapter 3 – Optimization Interface and Setup

Model Definition Structure

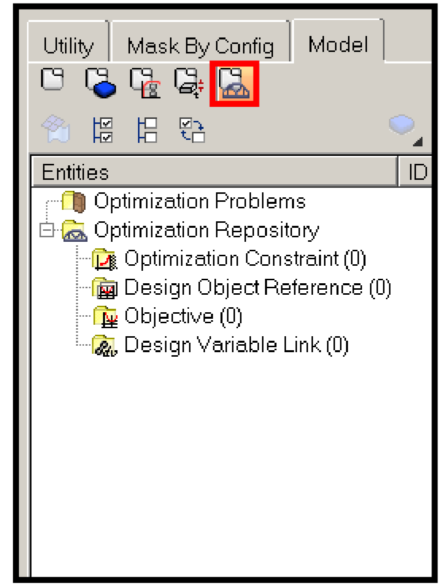
Optimization Setup

How to setup an optimization on HyperMesh

Optimization GUI

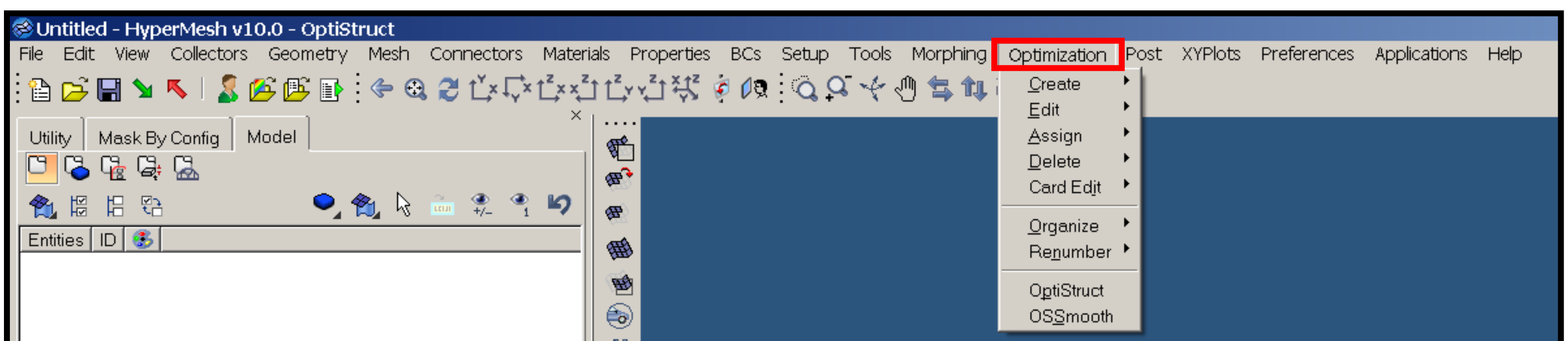


Optimization Panel



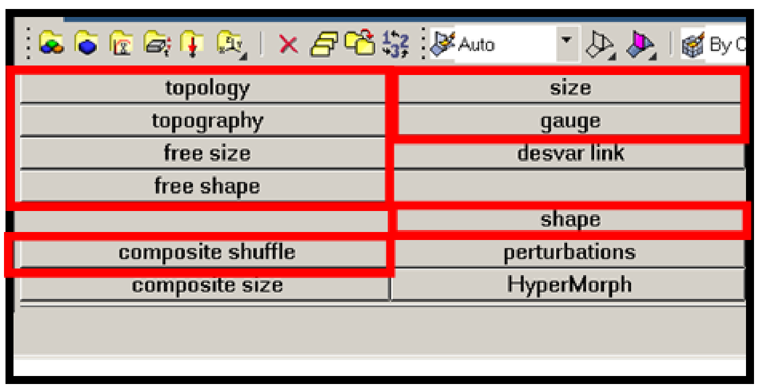
Model Browser

Optimization Menu

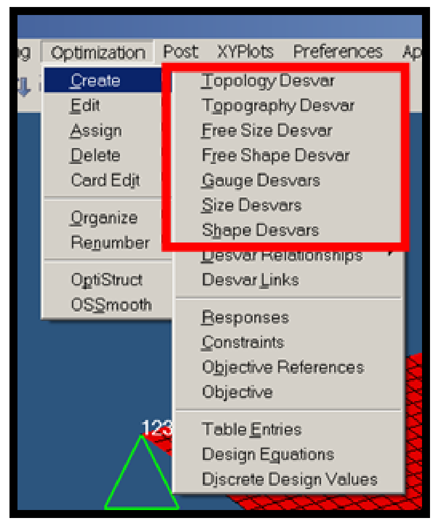


Optimization Setup module in HyperMesh

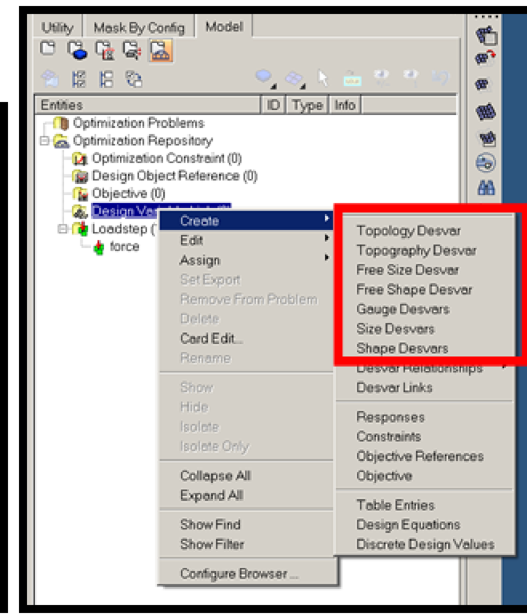
- Definition of Design Variables



Optimization panel



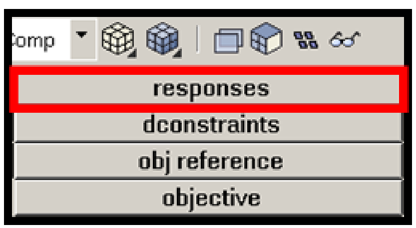
Optimization Menu



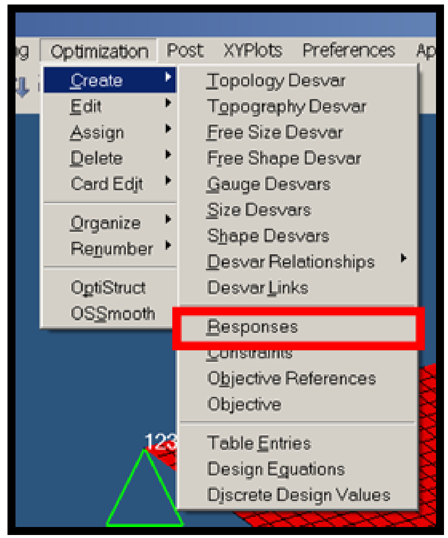
Model Browser

Optimization Setup module in HyperMesh

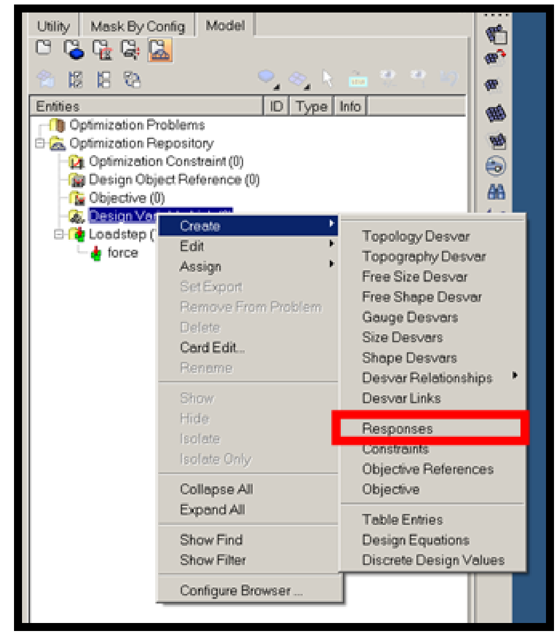
- Definition of Responses



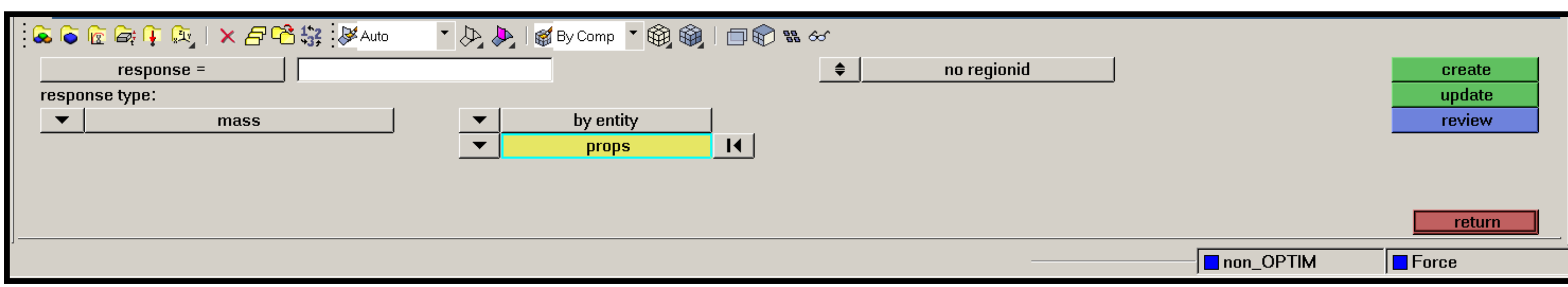
Optimization panel



Optimization Menu

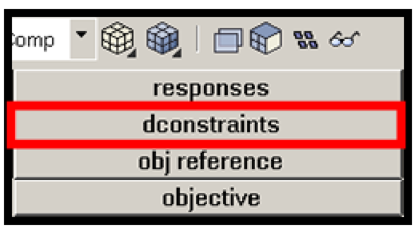


Model Browser



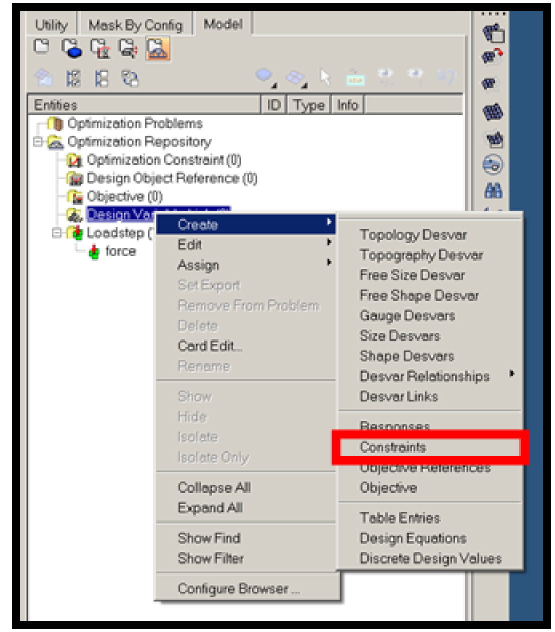
Optimization Setup module in HyperMesh

- Definition of Design Constraint

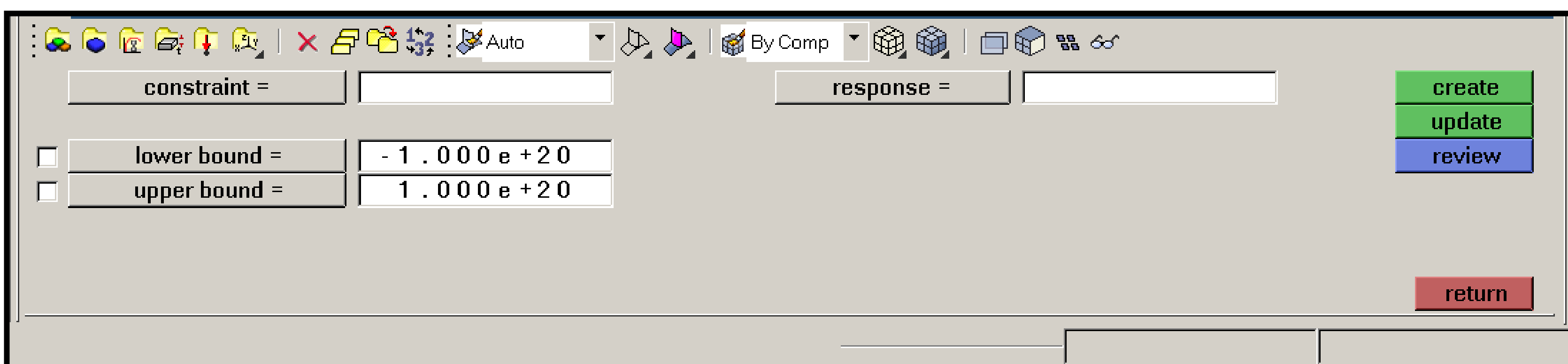


Optimization panel

Optimization Menu

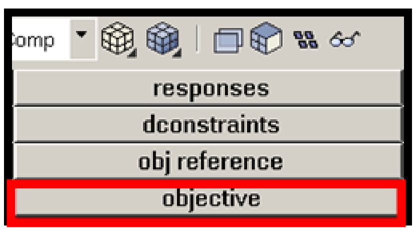


Model Browser

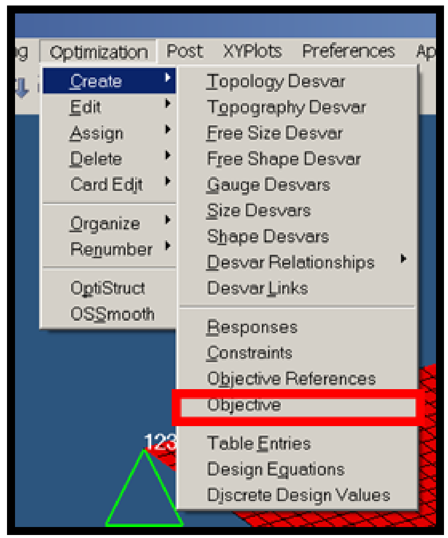


Optimization Setup module in HyperMesh

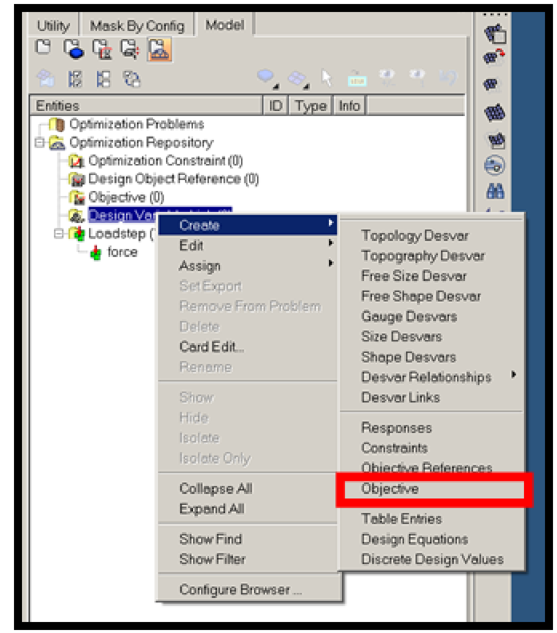
- Definition of Objective



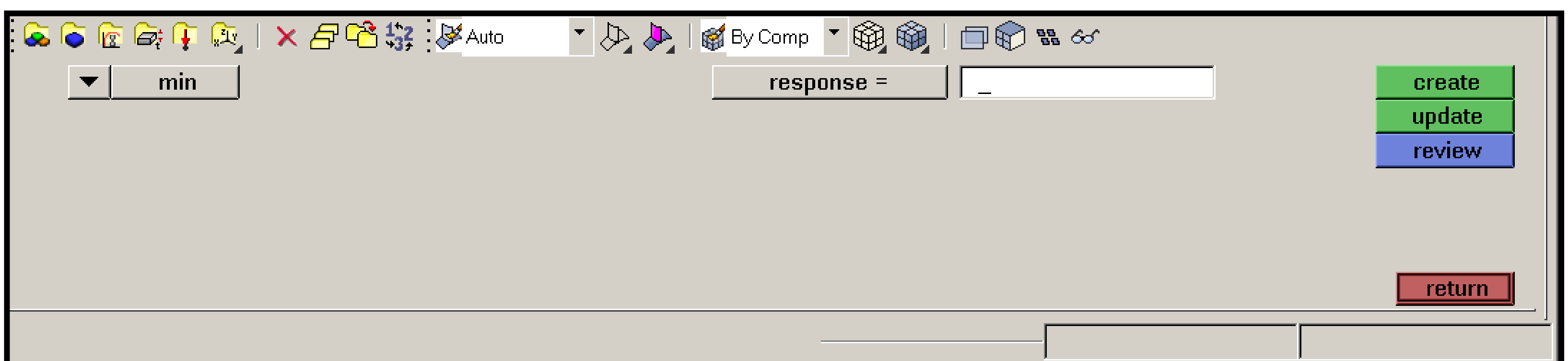
Optimization panel



Optimization Menu



Model Browser



Model Definition Structure

- Input/Output Section
- Subcase Information Section
  - Define Load Cases (Sub Cases, Load Steps)
  - Definition of Objective and Constraint Reference
- Bulk Data Section
  - Optimization Problem
    - ✓ Design Variables
    - ✓ Responses
    - ✓ Constraints
  - Optimization parameters (DOPTPRM)
  - Finite Element Model

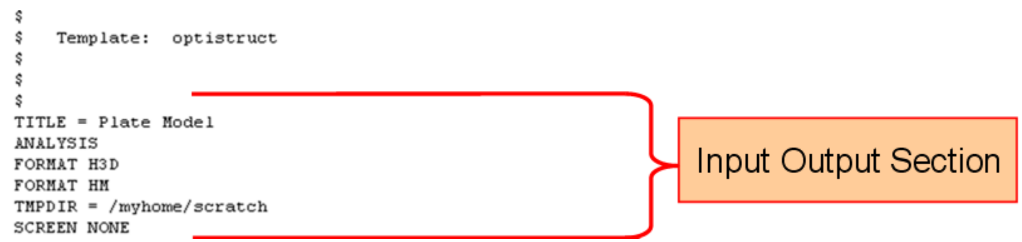
```
$
$   Template:  optistruct
$
$
$
TITLE = Plate Model
ANALYSIS
FORMAT H3D
FORMAT HM
TMPDIR = /myhome/scratch
SCREEN NONE
$------------------------------------------------------------------------------$
$                        Case Control Cards                                     $
$------------------------------------------------------------------------------$
$
$HMNAME LOADSTEPS        1Load
$
SUBCASE        1
  SPC  =        1
  LOAD =        1
$
BEGIN BULK

$
$  GRID Data
$
GRID          11        33.333330.0     0.0
GRID          12        31.296290.0     0.0
GRID          13        29.259250.0     0.0
...
...
$
$  CQUAD4 Elements
$
CQUAD4         1        2       74      11      12      75
CQUAD4         2        2       40      41      42      76
CQUAD4         3        2       76      42      43      77
...
...
```

Input Output Section

Case Control Section

Bulk Data Section

Model Definition Structure



- Input/Output Section
  1. ASCII output

     (.out ;.stat; .hist; .sh; .desvar; .prop; .hgdata; .grid; .oss; .HM.comp.cmf; .HM.ent.cmf)

  2. HTML Reports

     (.html ; _frames.html ; _menu.html; .shuf.html)

  3. Model results

     (.res; .h3d; _des.h3d; _s#.h3d

  4. HV session file

     (.mvw; _hist.mvw)

Model Definition Structure

- Optimization Cards

  - **Subcase Information Entry**

DESGLB	DESOBJ	DESSUB	DESVAR
MINMAX or MAXMIN	MODEWEIGHT	MODTRAK	NORM
REPGLB	REPSUB	WEIGHT	

  - **BULK Data Entry**

BEAD	BMFACE	DCOMP	DCONADD	DCONSTR
DDVAL	DEQATN	DESVAR	DLINK	DLINK2
DOBJREF	DOPTPRM	DREPADD	DREPORT	DRESP1
DRESP2	DRESP3	DSCREEN	DSHAPE	DSHUFFLE
DSIZE	DTABLE	DTPG	DTPL	DVGRID
DVMREL1	DVMREL2	DVPREL1	DVPREL2	

```
$------------------------------------------------------------------------------$
$                          Case Control Cards                                  $
$------------------------------------------------------------------------------$
$
$HMNAME LOADSTEPS       1Load
$
SUBCASE       1
  SPC  =       1
  LOAD =       1
$
BEGIN BULK

$
$  GRID Data
$
GRID          11        33.333330.0     0.0
GRID          12        31.296290.0     0.0
GRID          13        29.259250.0     0.0
...
...
$
$  CQUAD4 Elements
$
CQUAD4         1       2       74      11      12      75
CQUAD4         2       2       40      41      42      76
CQUAD4         3       2       76      42      43      77
...
...
```

Case Control Section

Bulk Data Section

The complete descriptions of these cards are available at the online documentation.

Constraint and Objective definition

- DCONSTR
  - Defines Responses as optimization constraints.
  - Relates response to lower and/or upper bound
- DCONADD
  - Adds constraints under same id
- DESSUB, DESGLB
  - Load case dependent, and independent reference in Case Control Section
- DESOBJ
  - Load case dependent, and independent reference in Case Control Section
  - Min/max

Optimization Cards

- DEQATN
  - Defines an equation
  - Linked to DVPREL2, DRESP2 for user defined property or response.
- DTABLE
  - Defines constants used in DEQATN
  - Linked to DVPREL2, DRESP2
- DSCREEN
  - Constraint screening definition
- DOPTPRM
  - Optimization parameter definitions
  - Max number of iterations, minimum member size control, moving limits, tolerances

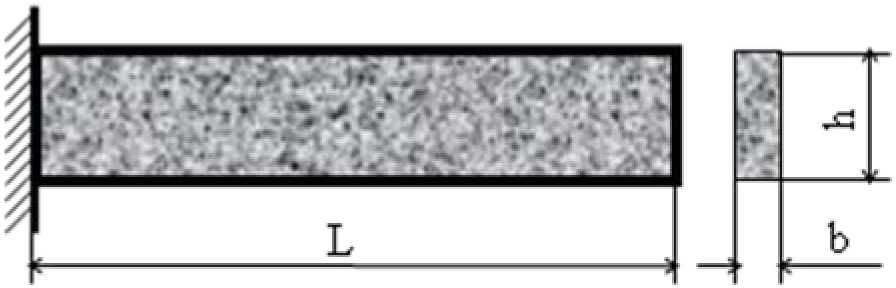
Constraint and Objective Definition: Load Case Reference

Objective and design constraints need to be defined load case dependent if the response is a reaction to a load

- Load case dependent
  - Compliance, frequency, displacement, stress, strain, force, composite responses
  - Functions using these responses w/o load case assignment
- Load case in-dependent (global)
  - Mass, mass fraction, volume, volume fraction, center of gravity, moments of inertia, weighted compliance, weighted frequency, compliance index
  - Functions using these responses
  - Functions using compliance, frequency, displacement, stress, strain, force, composite responses with load case assignment

Optimization Setup

- How to setup an optimization on HyperMesh



$$Min(f_1)$$
$$Mass \le 5.0E-04 \text{ ton}$$
$$5 \le \text{b} \le 15$$
$$5 \le \text{h} \le 15$$

- Geometry:
  - (L = 1000, h_0 = 10, b_0 = 10 mm)
- One load case: Normal Modes
  - First mode
- Material STEEL:
  - $\rho = 7.8e^{-9}$ t/mm³ [RHO] Density
  - E = 210000 MPa [E] Young's modulus
  - ν = 0.3 - [nu] Poisson's ratio

Optimization Setup

- How to setup an optimization on HyperMesh

Step 1 - Setup the Finite element analysis.

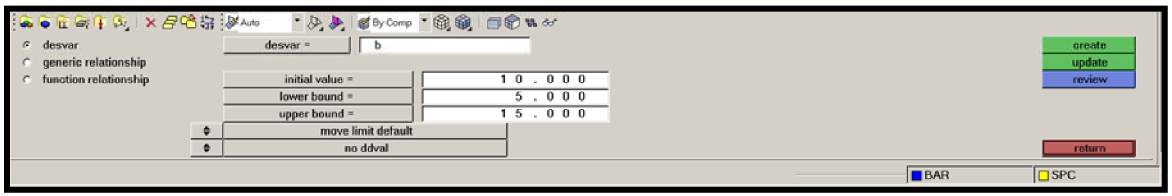
```
SUBCASE 1
  SPC = 1
  METHOD(STRUCTURE) = 2
BEGIN BULK
GRID           1        0.0     0.0     0.0
GRID           2        1000.0  0.0     0.0
CBAR           1       1       1       20.0
1.0     0.0
PBARL          1       1             BAR
+
+       10.0    10.0
MAT1           1210000.0        0.3     7.80E-09
EIGRL          2                       1
MASS
SPC            1       1  1234560.0
SPC            1       2  3     0.0
ENDDATA
```

Optimization Setup

- How to setup an optimization on HyperMesh

Step 2 - Define the Design Variables.

Optimization > Create > Size Desvars




```
DESVAR         1     b10.0    5.0     15.0
DESVAR         2     c10.0    5.0     15.0
```

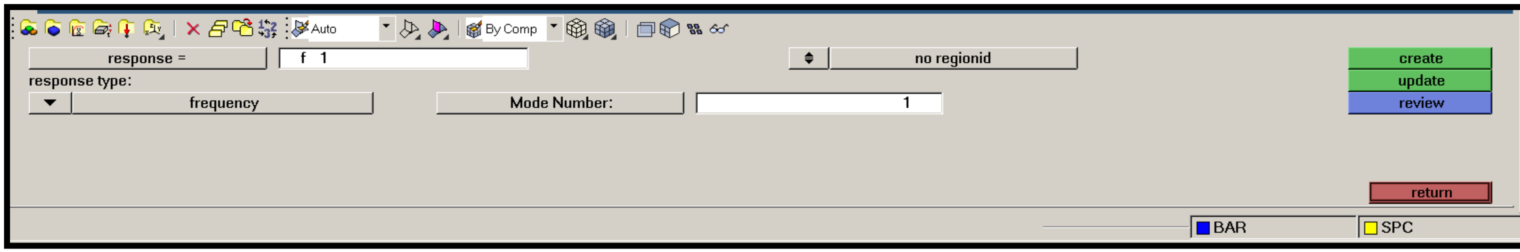

```
DVPREL1 1       PBARL           1DIM1                                   0.0
+       1       1.0
DVPREL1 2       PBARL           1DIM2                                   0.0
+       2       1.0
```

Optimization Setup

- How to setup an optimization on HyperMesh

Step 3 - Define the Responses.

Optimization > Create > Response



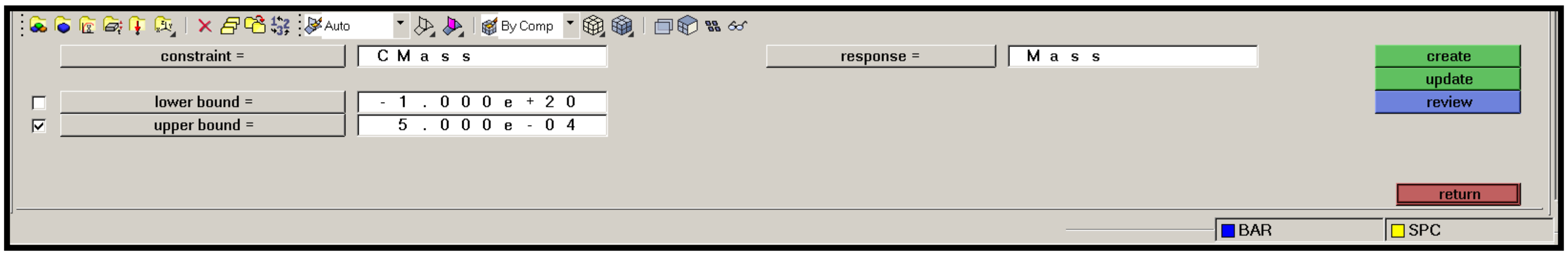

```
DRESP1  1       f1      FREQ                            1
DRESP1  2       Mass    MASS
```

Optimization Setup

- How to setup an optimization on HyperMesh

Step 4 - Define the constraints.

Optimization > Create > Constraints



This creates on the Subcase Information section:

```
DESGLB          2
```

This creates on the bulk data section:

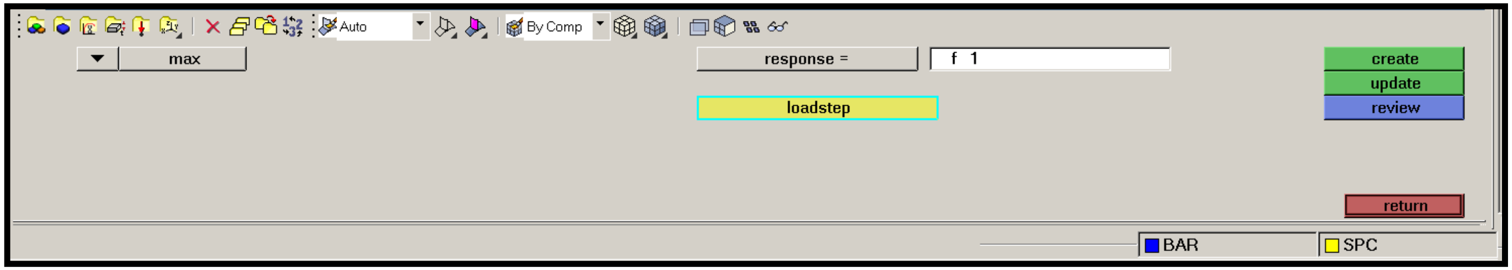
```
DCONSTR         1          2          5.00E-04
DCONADD         2          1
```

Optimization Setup

- How to setup an optimization on HyperMesh

Step 5 - Define the Objective

Optimization > Create > Objective



This creates on the Subcase Information section:

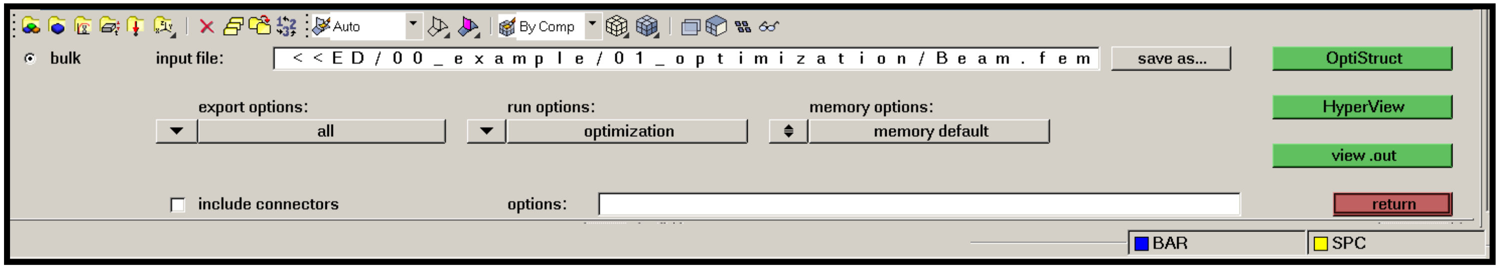
```
DESOBJ(MAX)=1
```

Optimization Setup

- How to setup an optimization on HyperMesh

Step 6 - Run the Simulation

Application > OptiStruct



Optimization Setup

- How to setup an optimization on HyperMesh

FINAL SETUP

```
DESGLB         2
SUBCASE 1
  SPC = 1
  METHOD(STRUCTURE) = 2
DESOBJ(MAX)=1
BEGIN BULK
DESVAR         1       b10.0   5.0     15.0
DESVAR         2       c10.0   5.0     15.0
DVPREL1 1      PBARL           1DIM1                   0.0
+       1      1.0
DVPREL1 2      PBARL           1DIM2                   0.0
+       2      1.0
DRESP1  1      f1      FREQ                           1
DRESP1  2      Mass    MASS
DCONSTR        1       2       5.00E-04
DCONADD        2       1
GRID           1        0.0    0.0     0.0
GRID           2        1000.0 0.0     0.0
CBAR           1       1       1     20.0    1.0     0.0
PBARL          1       1            BAR
+
+       10.0   10.0
MAT1           1210000.0       0.3     7.80E-09
EIGRL          2                    1
MASS
SPC            1       1  1234560.0
SPC            1       2  3    0.0
ENDDATA
```

Chapter 4 – Concept Design

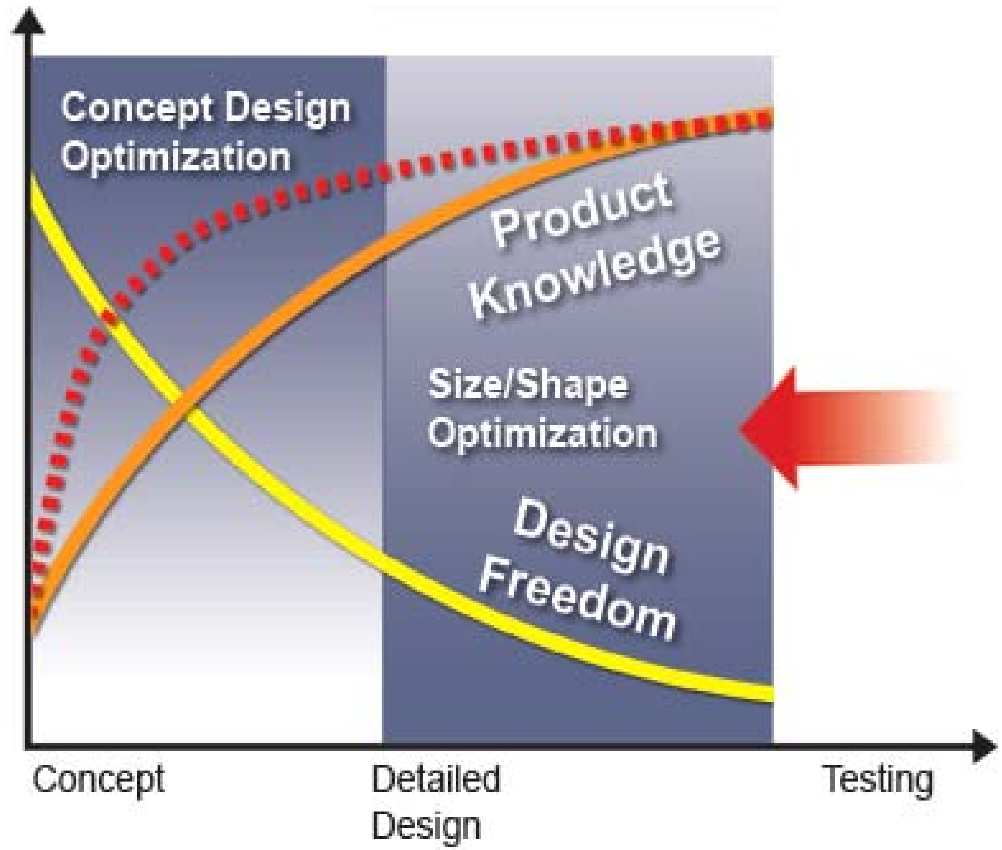
Topology Optimization

Topography Optimization

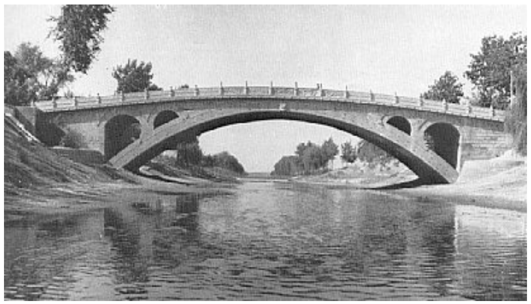
Free-size Optimization

How Structural Optimization Cuts Development Time

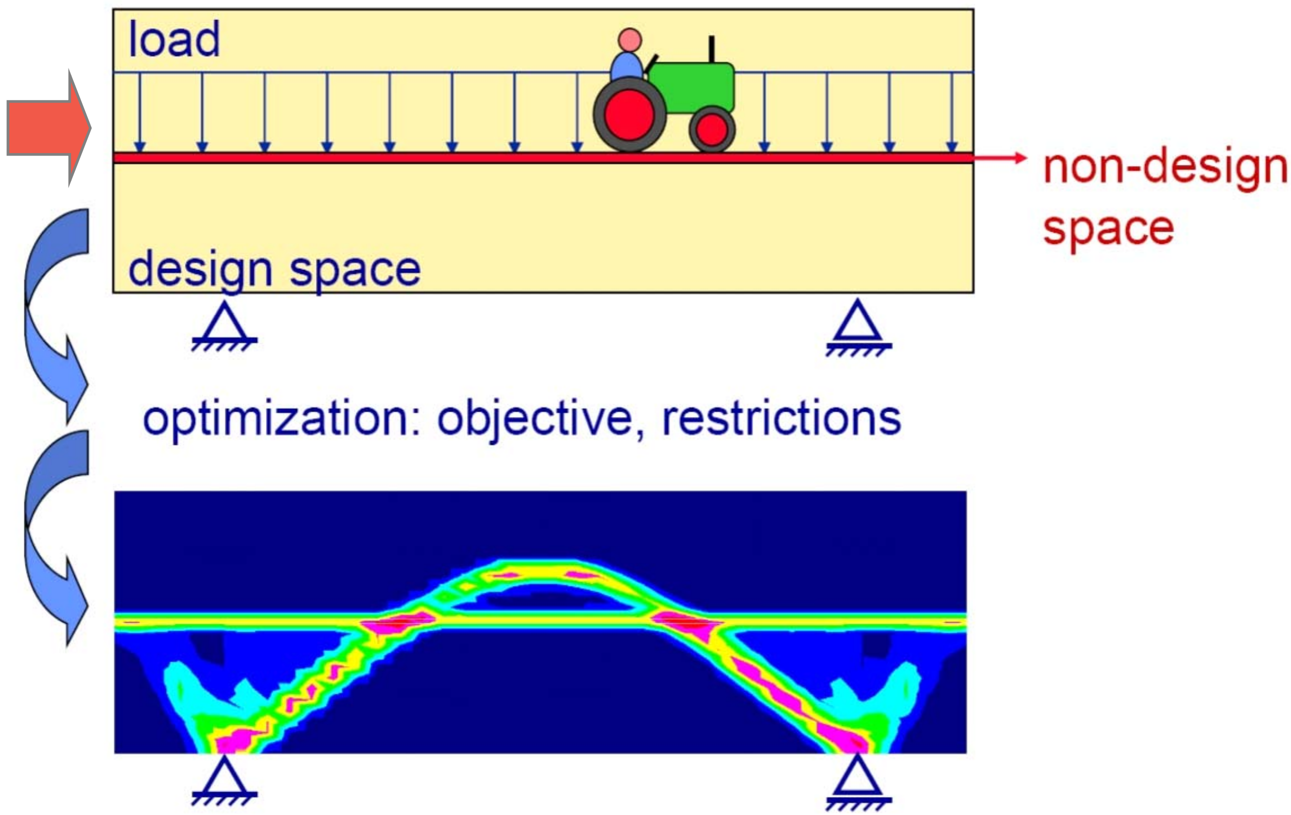
- Most of the product cost is determined at the concept design stage
- **Problem:**
  minimum knowledge, but maximum freedom
- **Need:**
  effective concept design tools to minimize downstream "re-design" costs and time-to-market



Topology Optimization



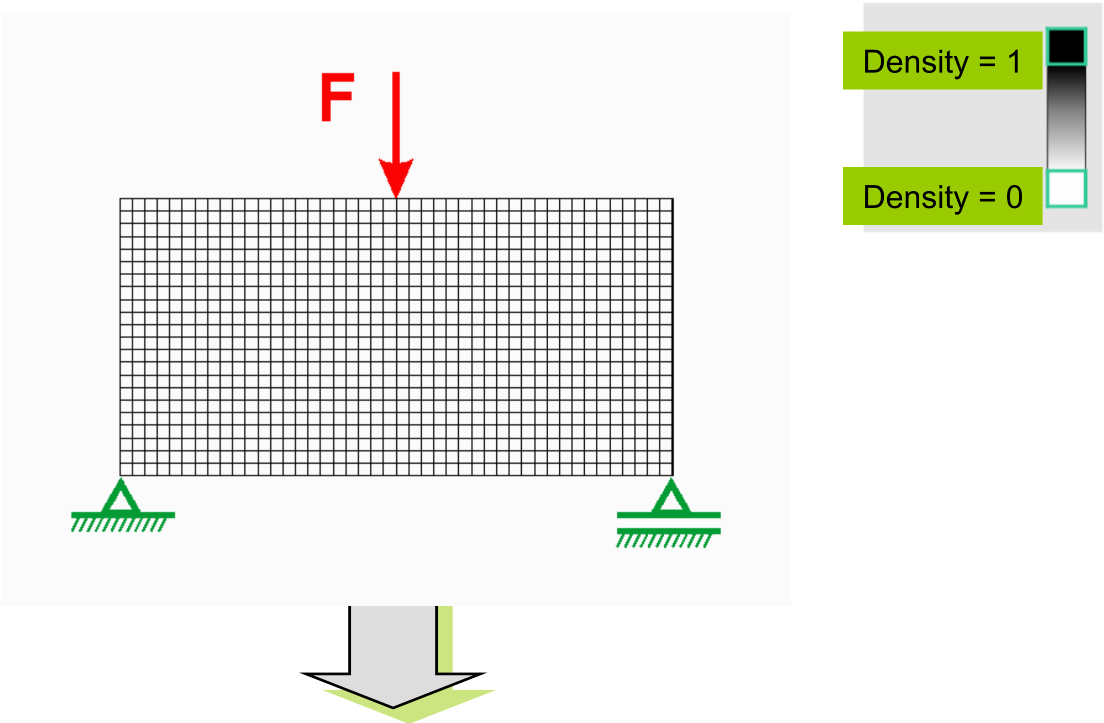
Baseline design

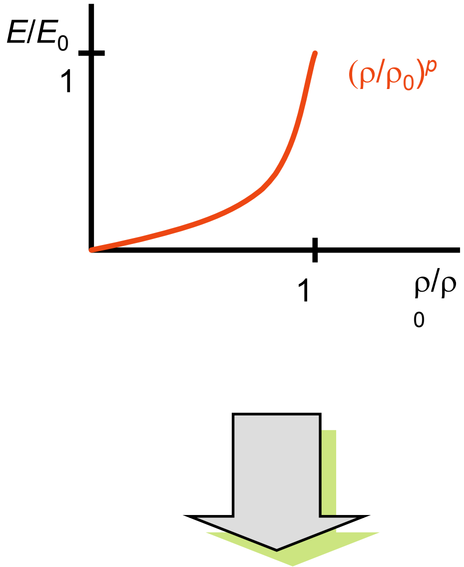


Altair® OptiStruct® *design proposal*

Design Variables Topology Optimization

What does OptiStruct change?





Density Method

Very robust

Penalty Factor

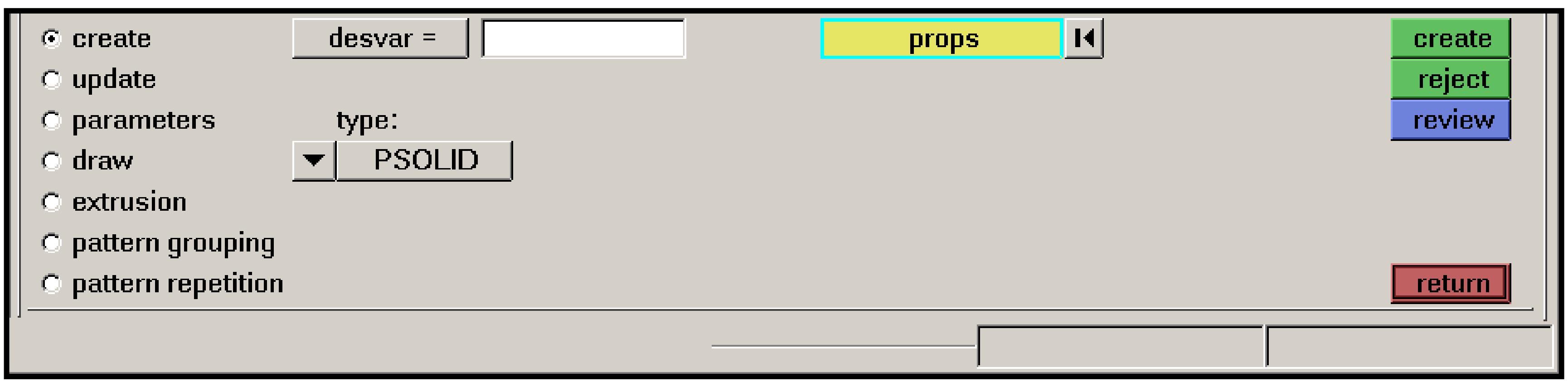
More discrete design proposals

OptiStruct Input: Topology Optimization

DTPL card – Design Variable definition for topology optimization

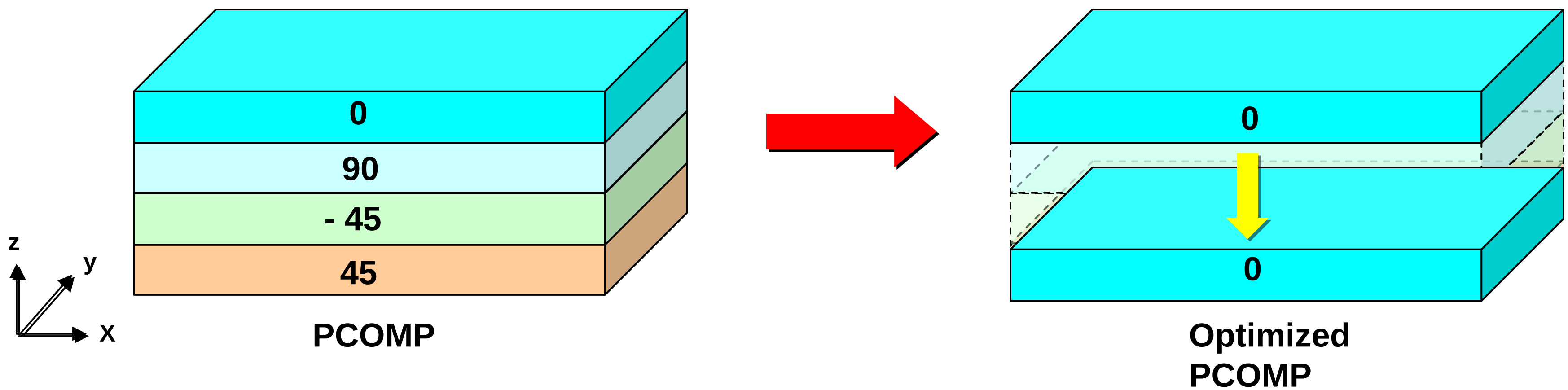
- Shells - Property with base and total thickness defines topology design space
- Solids – Properties define topology design space
- Composites (PCOMP) - Properties define topology design space
- Rod, Bar, Weld , Bush- Properties define topology design space
- Stress constraints bounds
- Manufacturing constraints definition

HyperMesh Topology panel:



Topology optimization on PCOMP

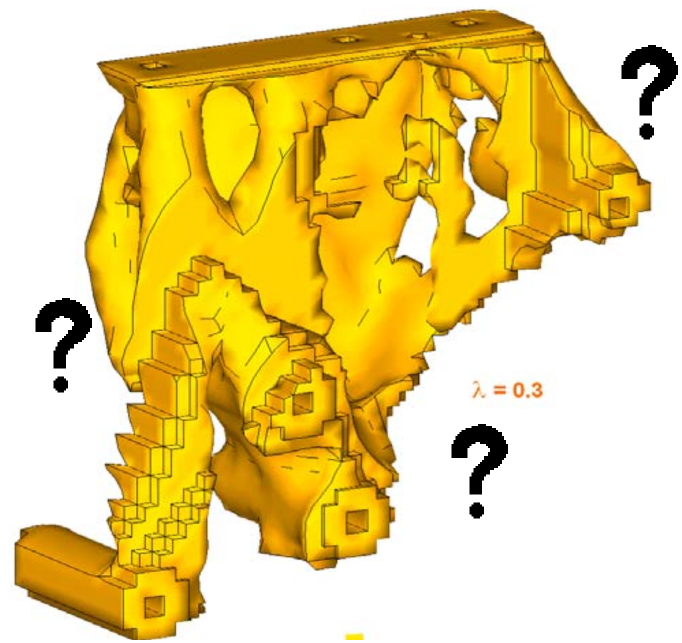
- Increase/decrease the thickness of given ply angle
- Ability to optimize the angle as well by creating “phantom” ply



- mat option on DTPL
  - Ply → ply based PCOMP (default)
  - Homo → homogenized PSHELL

Topology Optimization using Manufacturing Constraints

- What are Manufacturing Constraints?
  - Additional input for the optimization problem
  - OptiStruct tries to meet manufacturing constraints
- Why are they so important?
  - Make it much easier to interpret optimization results
  - Use of standard profiles/manufacturing tools/processes
  - Optimized structures are of no value if nobody can manufacture them
- Implemented manufacturing constraints
  - Maximum member size
  - Minimum member size
  - Draw direction constraint
  - Pattern repetition
  - Pattern grouping
  - Extrusion constraint



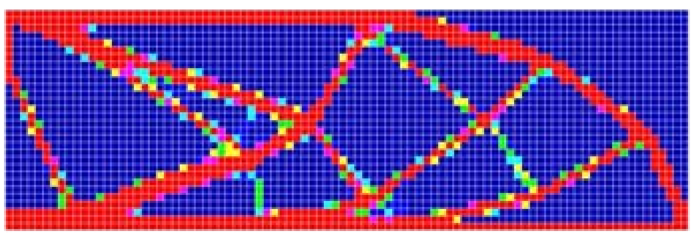
Topology Optimization using Manufacturing Constraints

Manufacturing constraints for topology optimization helps generate practical design concepts

- Minimum member size control specifies the smallest dimension to be retained in topology design. Controls checker board effect and discreteness.
- Maximum member size control specifies the largest dimension allowed in the topology design. It prevents large formation of large members and large material concentrations are forced to more discrete forms.
- Pattern grouping / repetition can be applied to enforce a repeating pattern or symmetrical design even if the loads applied on the structure are unsymmetrical or non-repeating.
- Draw direction / extrusion constraints can be applied to obtain design suitable for casting or machining operations by preventing undercut or die-lock cavities.

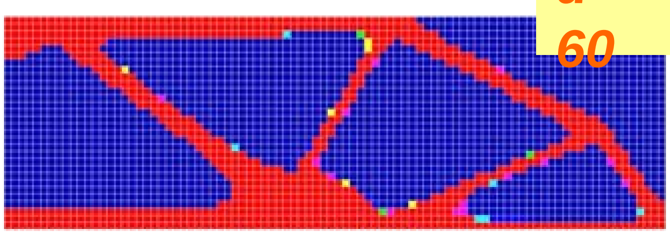
Manufacturing Constraints: Minimum Member Size Control

- Input: approximate minimum diameter *d* in two dimensions
- Works in 2D and 3D
- Controls the size of small structural features
- Controls "checkerboarding"
- Easier interpretation of the resulting layout
- Higher computation cost

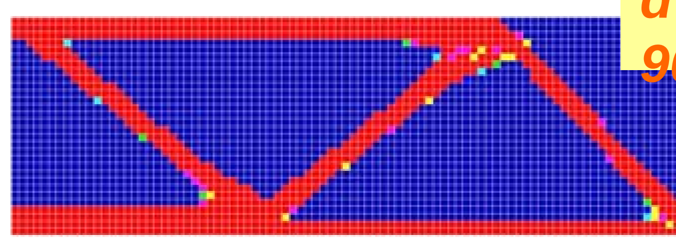


Without min member size

- *Difficult to manufacture due to micro structures*
- *Results are mesh dependent*



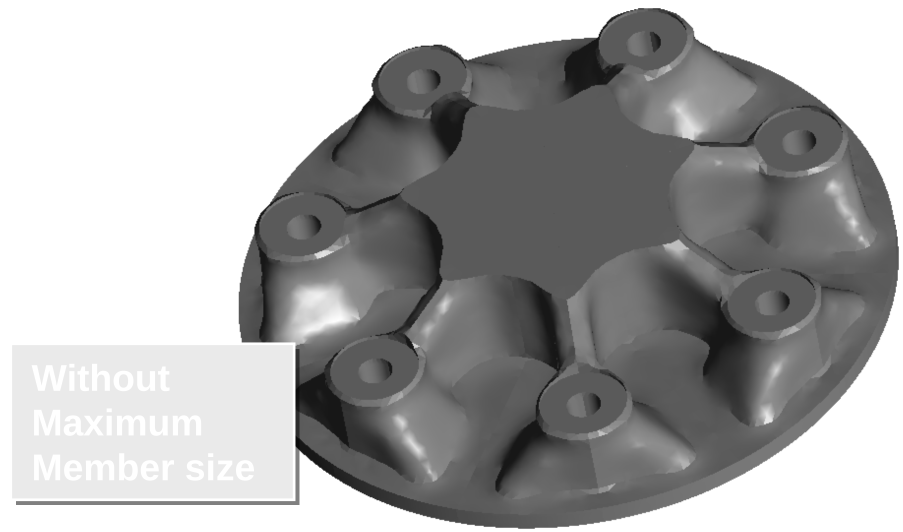
***d* = 60**

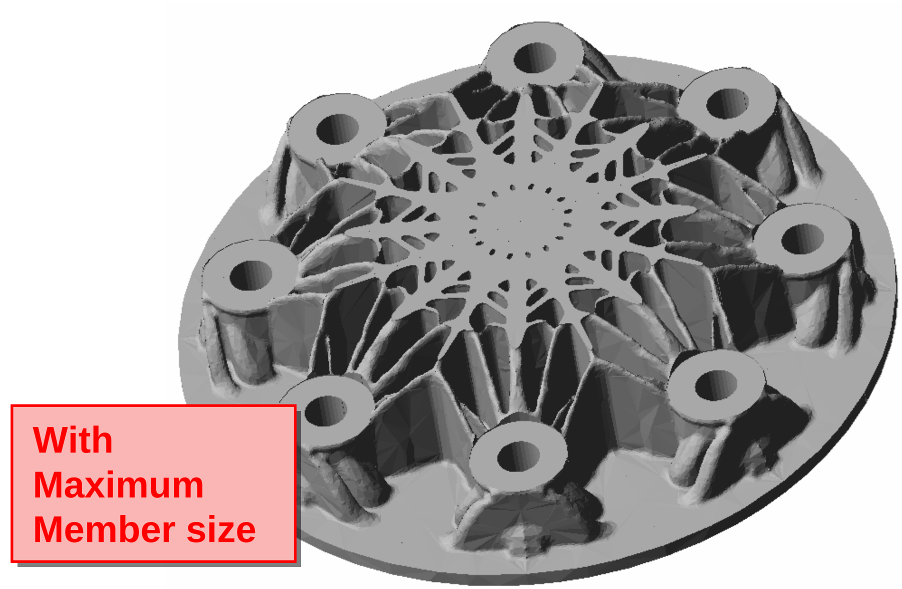


***d* = 90**

Manufacturing Constraints: Maximum Member Size Control

- Definition of maximum allowable structural member size
- Eliminates material concentrations
- Mesh considerations
  - Shell and solid elements
  - Tetrahedral and hexhedral
  - Min member > 3 X mesh size
  - Max member > 2 X min size





Manufacturing Constraints: Pattern Repetition

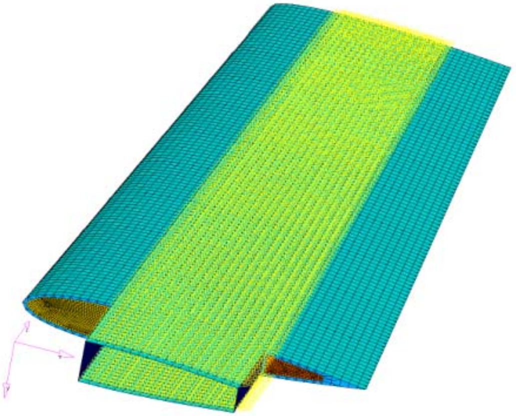
Cyclic Repetition

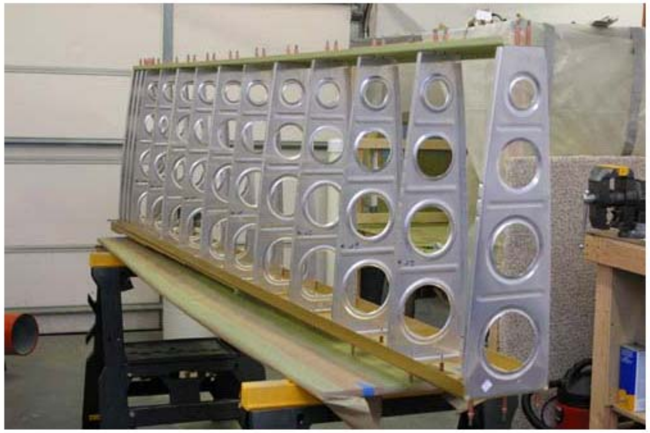
- Symmetry definitions
- Cyclic repetition of design features within a single domain
- User enters # of wedges
- Application: Cyclic structures with non symmetrical loadcases

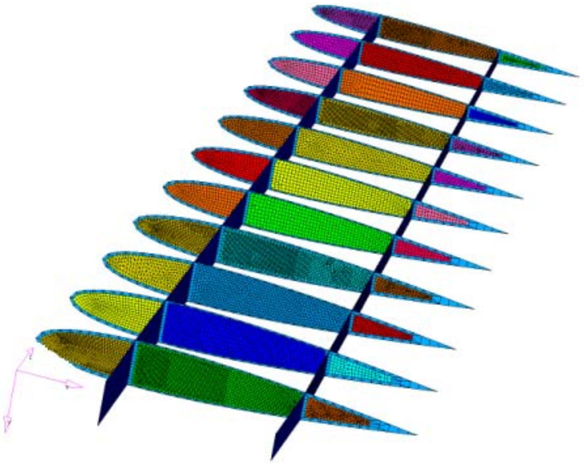
Pattern Repetition

Application example: Airplane Wing Ribs

- Goal: same topology on every rib
- Scaling factor to account for different sizes of design space

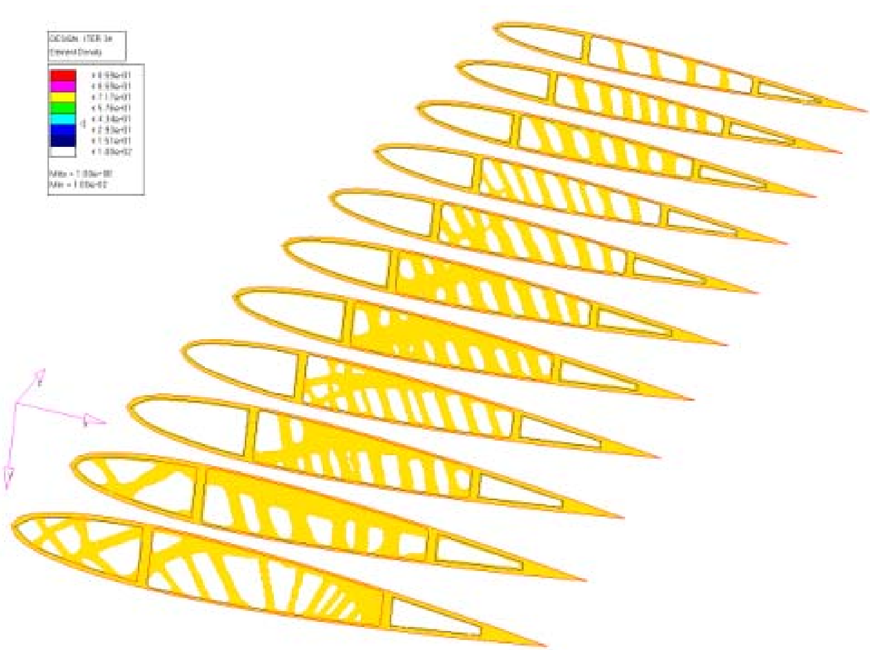




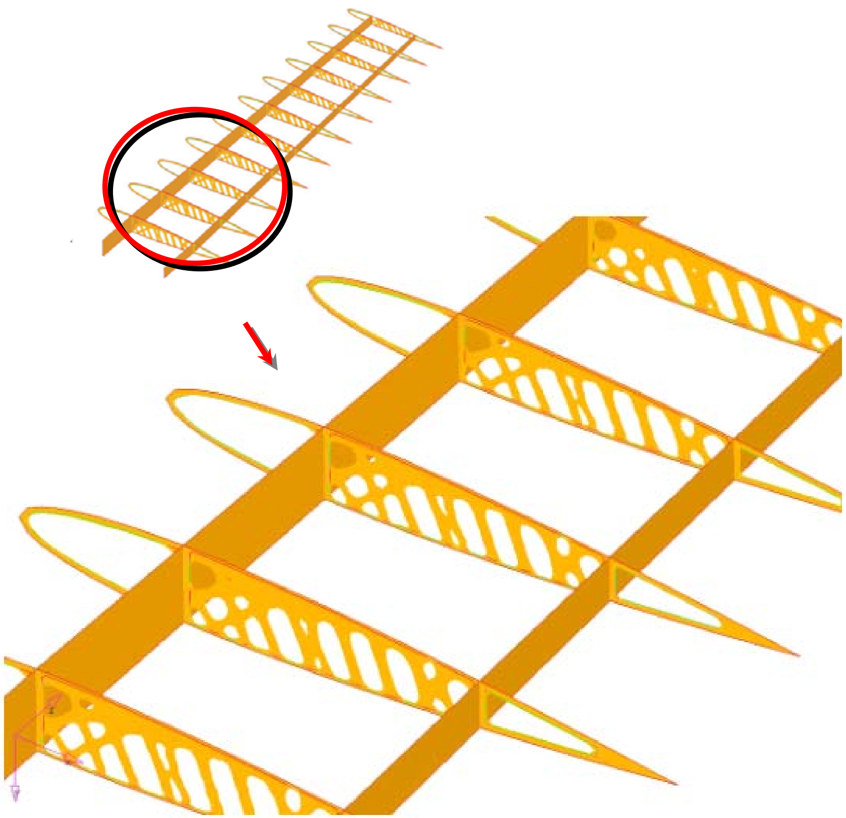


Pattern Repetition

Application example: Airplane Wing Ribs



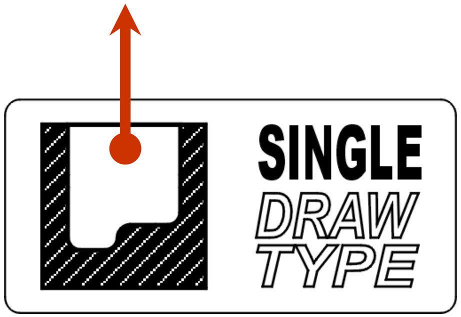
Without pattern repetition

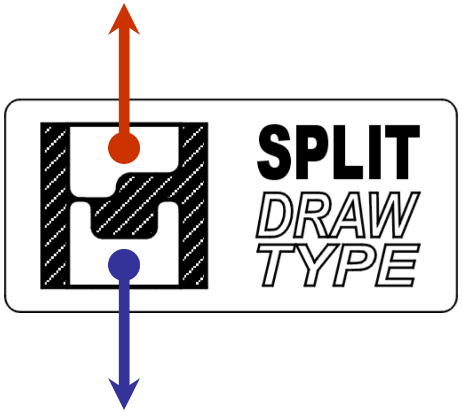


With pattern repetition

Draw Direction Constraint

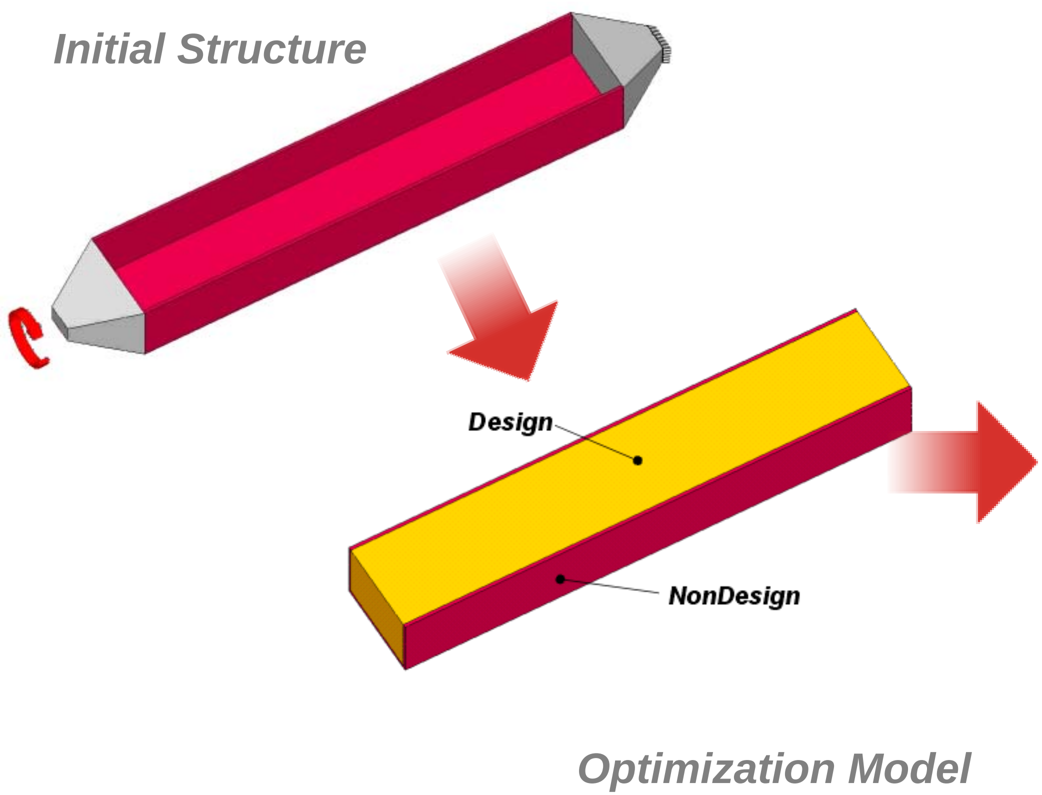
- Define global casting direction
- Eliminates undercuts in design proposal
- Reduces interpretation effort
- Important if part shall be manufactured by
  - Casting
  - Injection molding
  - Milling
- Draw type options
  - Single
  - Split



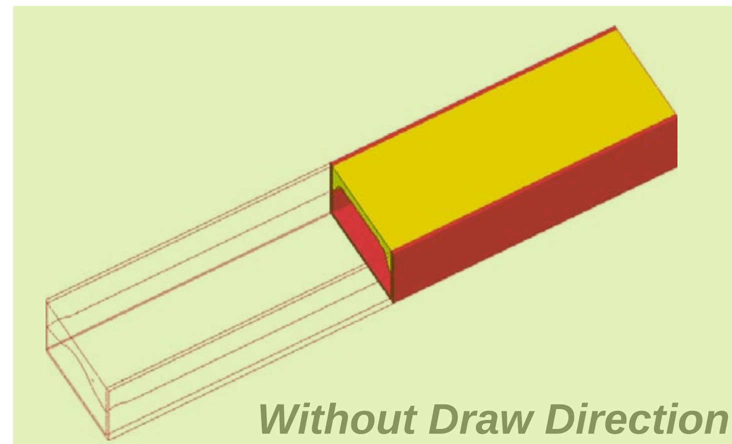


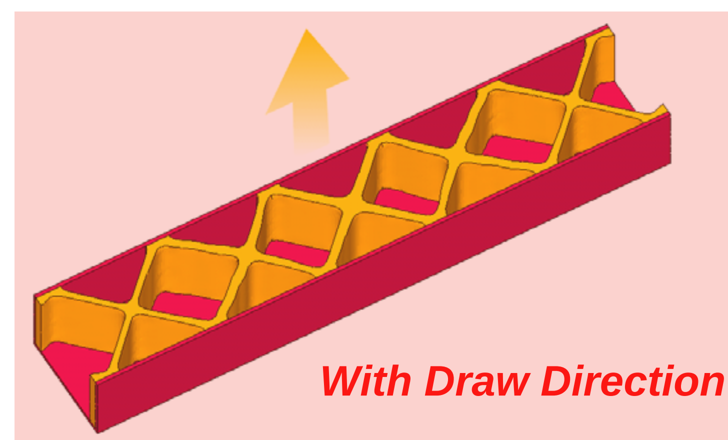
Draw Direction Constraint

Example: Determine Optimum Stiffeners in Torsion Loaded U-Profile



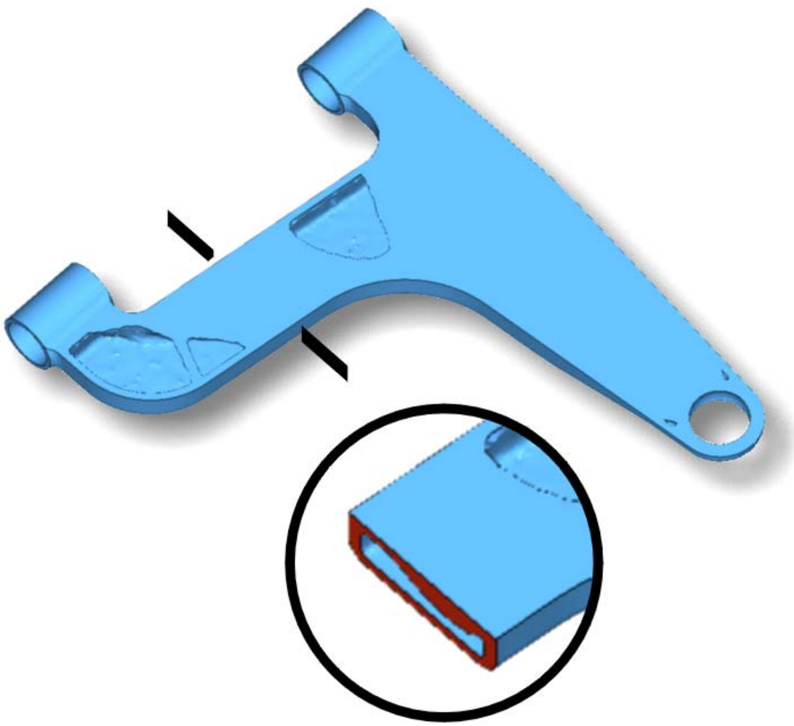
Optimization Results



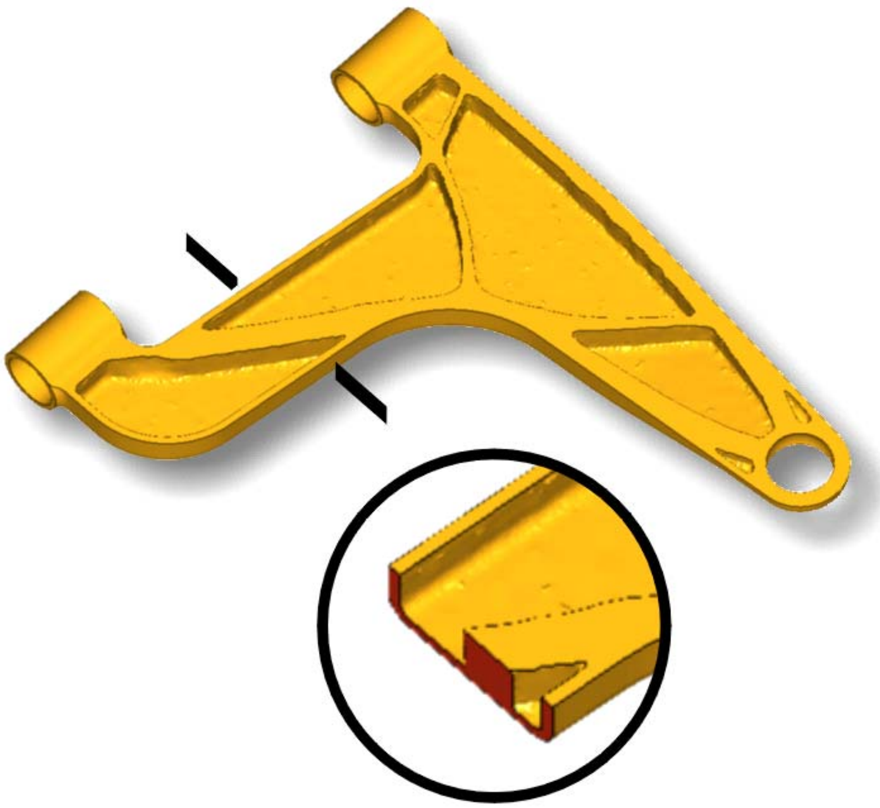


Draw Direction Constraint

Example: Optimum Rib Pattern of a Control Arm



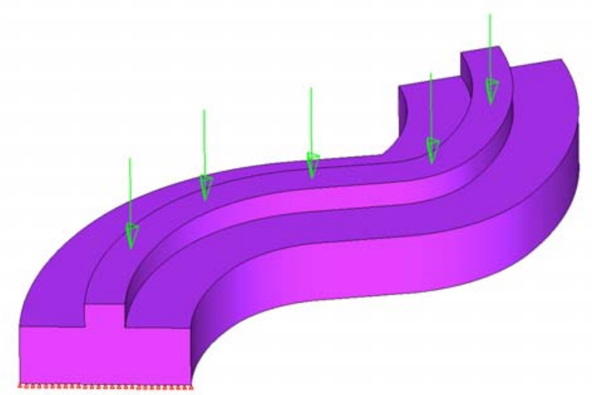
Without Draw Direction



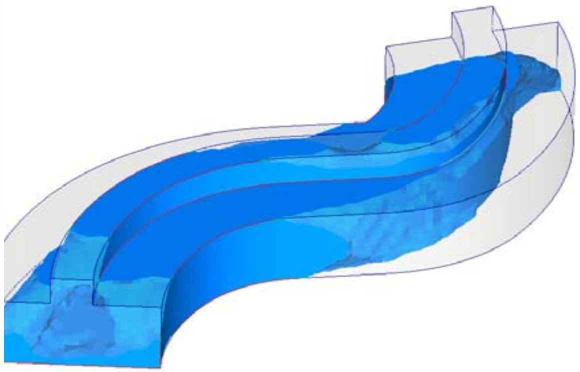
With Draw Direction

Extrusion Constraint

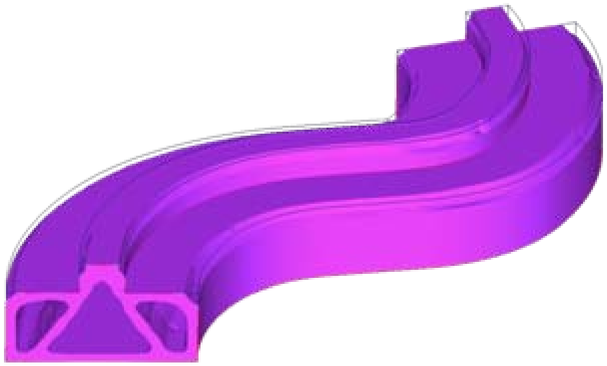
Manufacturing control for constant cross sections



Package space

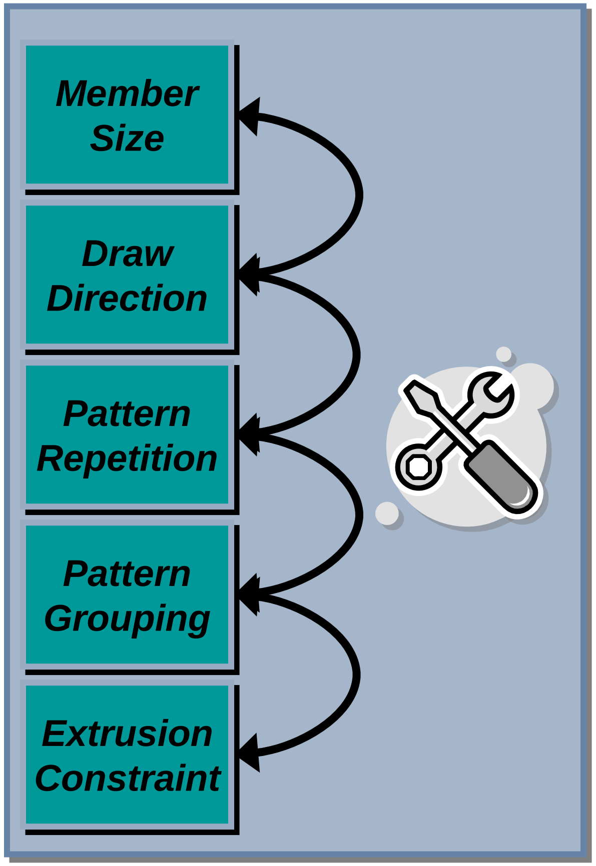


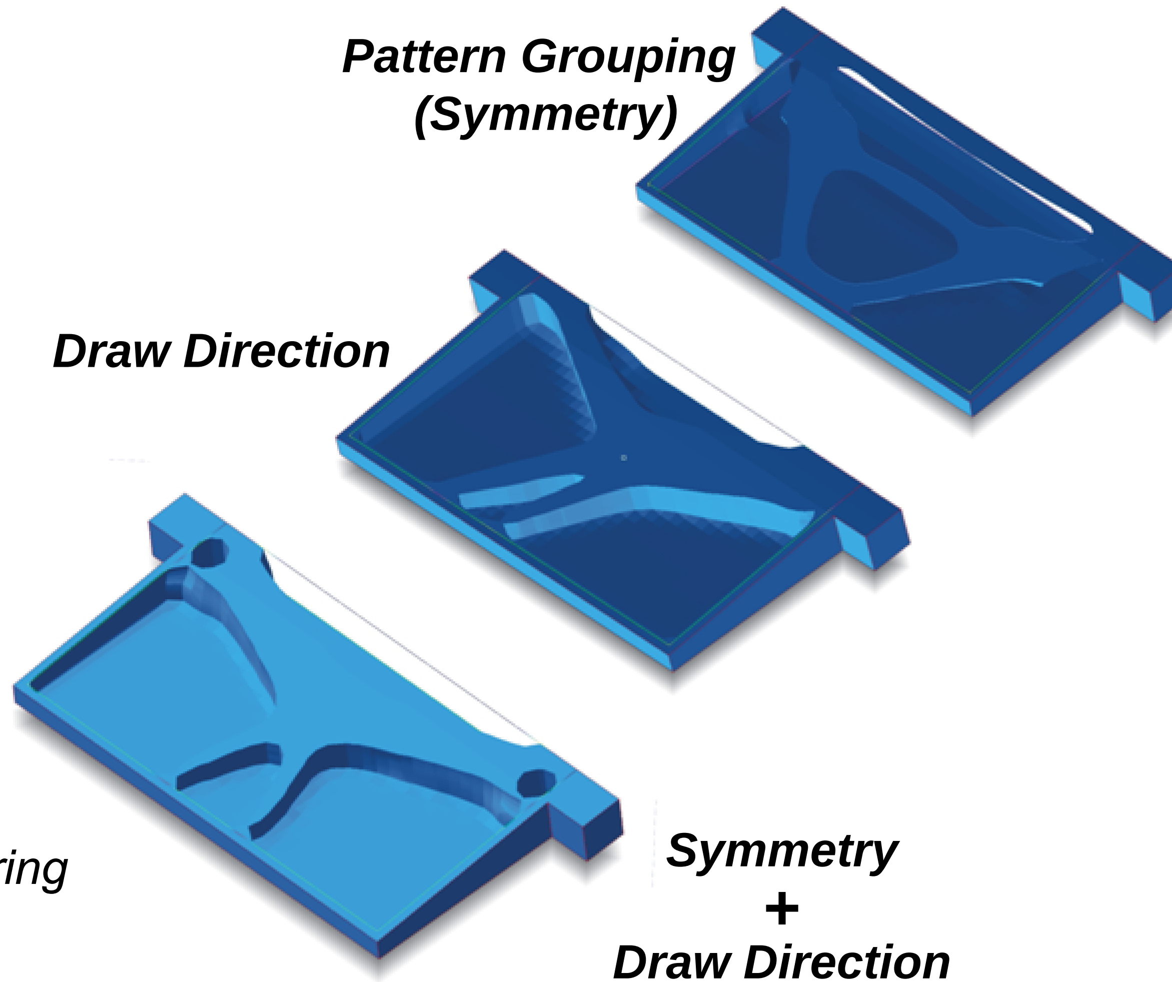
Design proposal without extrusion constraint



Design proposal with extrusion constraints

Combination of Manufacturing Constraints



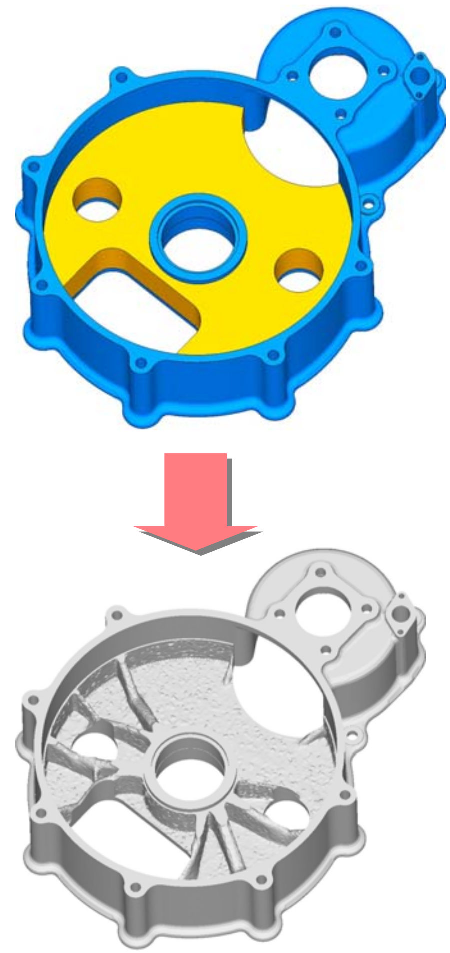


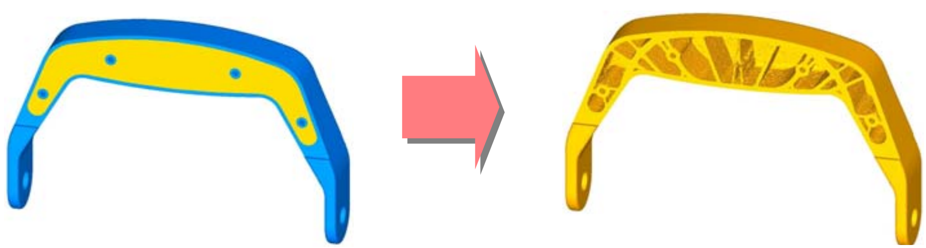
Any combination of manufacturing constraints is possible

Applications of Topology Optimization

Determination of Optimum Rib Patterns for Reinforcement

- Non design space represents general geometry concept
- Design space defines areas where ribs shall be introduced
- Manufacturing constraints crucial
  - Draw direction
  - Minimum & maximum member size





Common Topology Optimization Problems

- Minimize (weighted / total / regional) compliance

  with constrained (total / regional) volume / mass fraction

- Minimize (total / regional) volume/ mass fraction

  with constrained displacements

- Maximize (weighted) frequency

  with constrained (total / regional) volume / mass fraction

- Minimize (total / regional) volume / mass fraction

  with constrained frequencies

- Minimize combined compliance and frequencies

  with constrained (total / regional) volume / mass fraction

- Minimize (total / regional) volume/ mass fraction

  with stress constraints

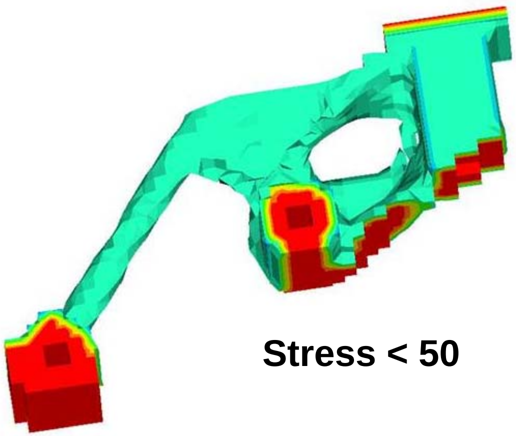
Additional Optimization Considerations

Constraint Screening (DSCREEN)

- Screening - specify normalized threshold value
  - *Temporarily ignores* constraints which are less than the normalized threshold value during optimization
- Regionalization - specify maximum number of constraints to be retained for a given region
  - Considers user specified number of most violated constraints for each load case and region id.
- Essential in situations where there are many constraints
  - E.g. Stress constraints for shape/size optimization.
- If too many constrained responses are screened, it may take considerably longer to reach a converged solution or, in the worst case, it may not be able to converge on a solution if the number of retained responses is less than the number of active constraints for the given problem.

Topology Optimization with stress Constraints

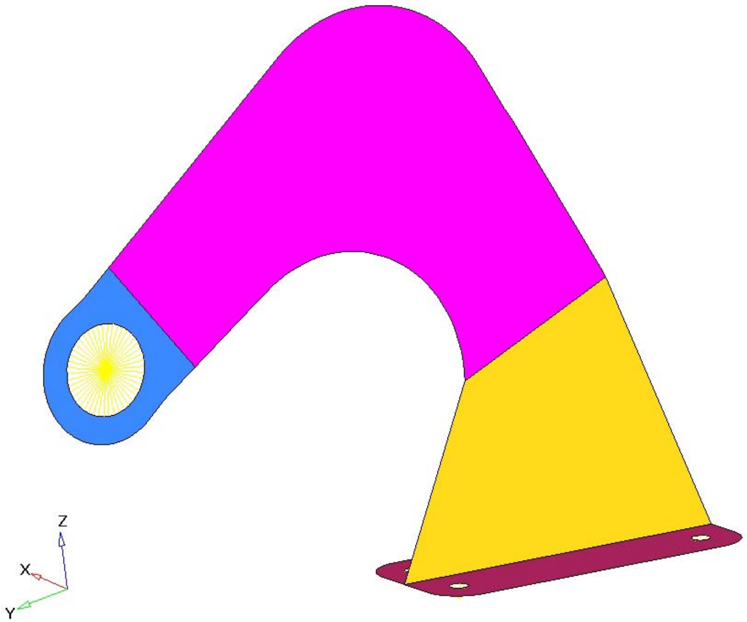
- Global von mises stress constraints
  - Apply to entire model including non design space
- Stress constraints for a partial domain of the structure are not allowed
  - The reason is that it often creates an ill-posed optimization problem as elimination of the partial domain would remove all stress constraints
- Local stresses are still high
  - This is for general stress level control
  - Local stress should be taken care of by using shape/size



Stress < 50

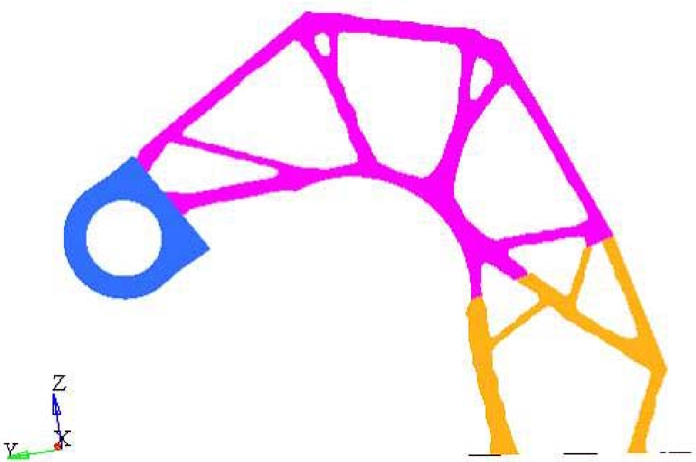
Stress < 30

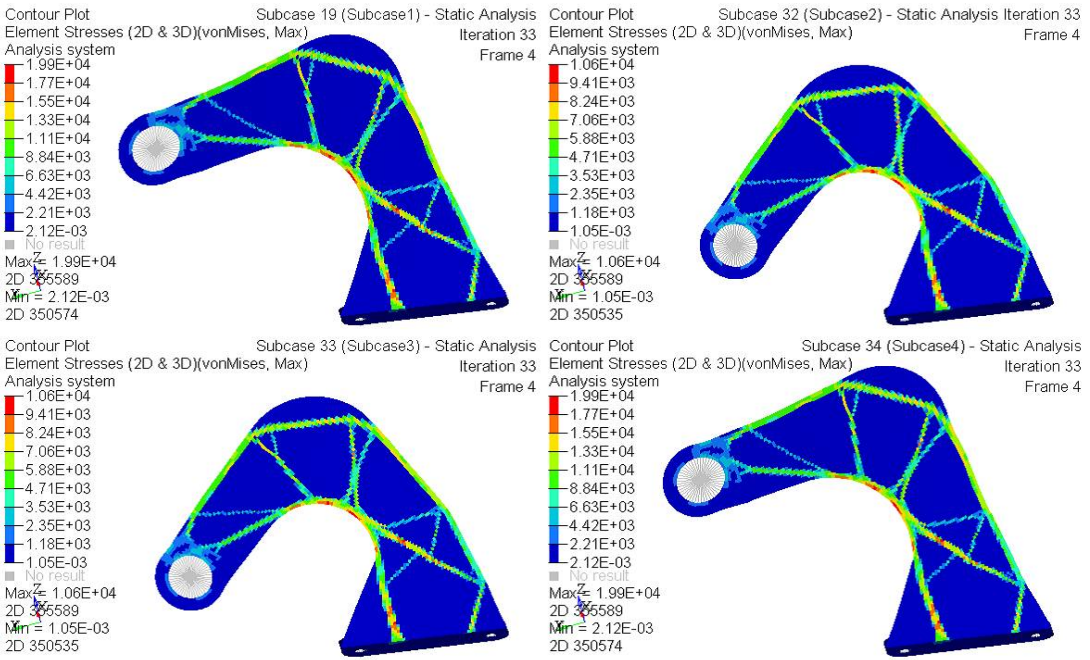
In this Exercise, a topology optimization is performed on a bracket-hook modeled with shell elements.



Objective function:	Minimize volume.
Constraints:	Von Mises stress < 1.6 e 04.
Design Variables:	The density of each element in the design space.

Exercise 4.1: Topology Optimization of a Hook with Stress Constraints



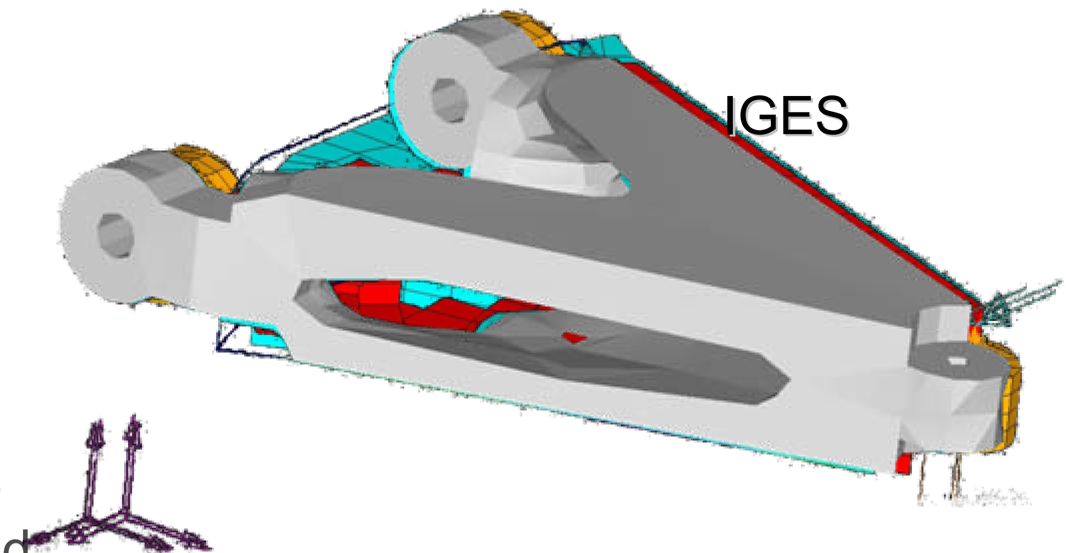


Stress results for all static sub case (Von Mises < 1.6e4)

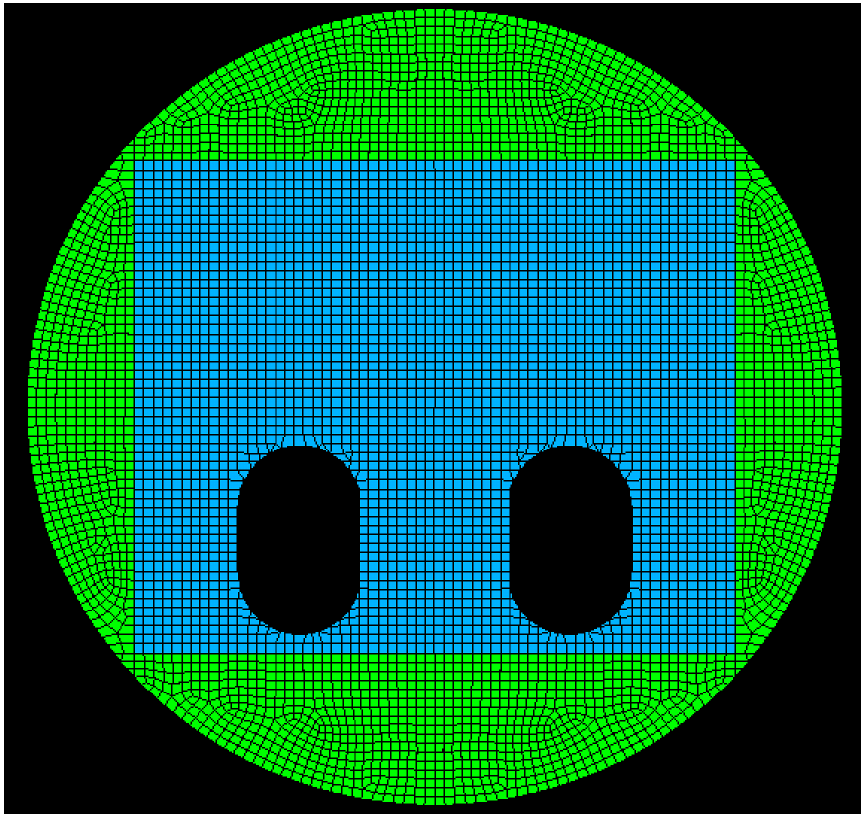
Notes:	The advantages of using stress based optimization over the classical minimize (compliance) subject to volume fraction constraint is that it eliminates the guessing of the right volume fraction. Additionally, it eliminates the need for compliance weighting bias for multiple subcases. There might still be high local stress regions which can be improved more effectively with local shape and size optimization.

OSSmooth: Geometry Extraction of Optimization Results

- A Geometry creation tool for Topology/Topography/Shape Optimized models
- Supports different output formats (IGES, STL, H3D etc.)
- Advanced geometry smoothing options for smoother surfaces
- Surface reduction option to reduce the size of IGES and STL files
- Integrated into HyperMesh and is easy to use



Topology Optimization Example: Bulkhead Stiffeners



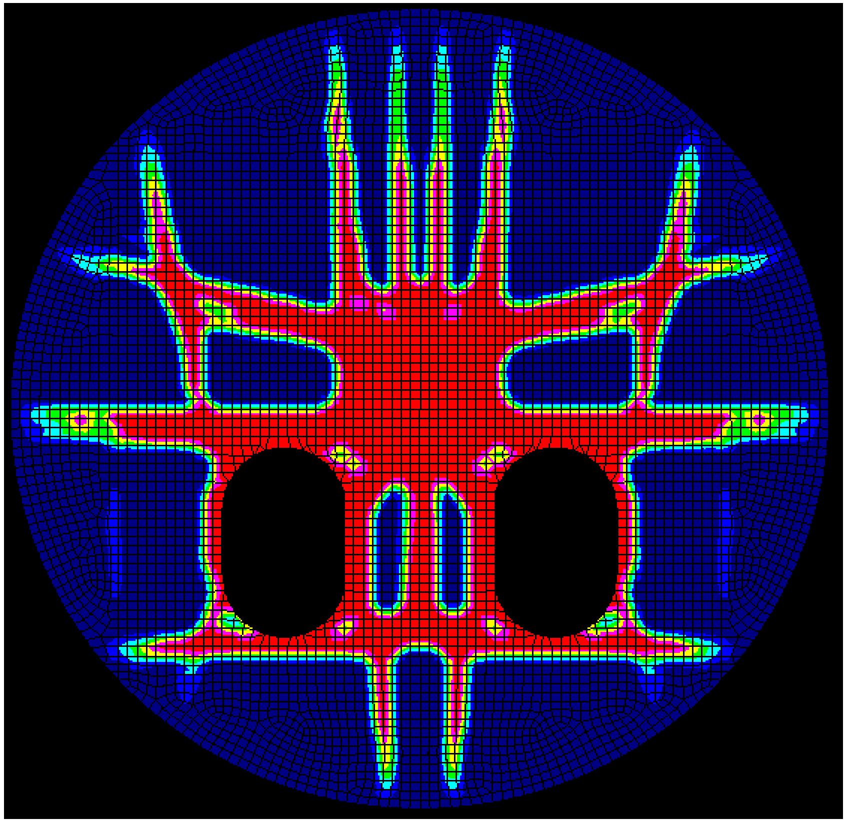
Design space

- Task: Stiffening of a bulk head using ribs
- 2 load cases
  - Hydrostatic load (fuel)
  - Take-off

    Pressure load on blue part
- Clamped perimeter
- 2 man holes

Topology Optimization Example: Bulkhead Stiffeners

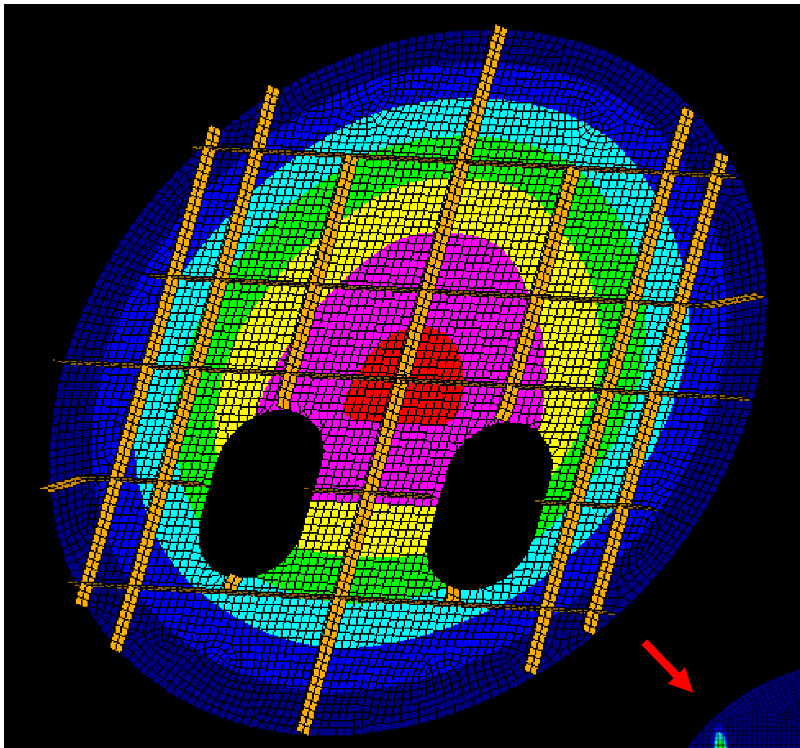
Topology Results



Stiffening

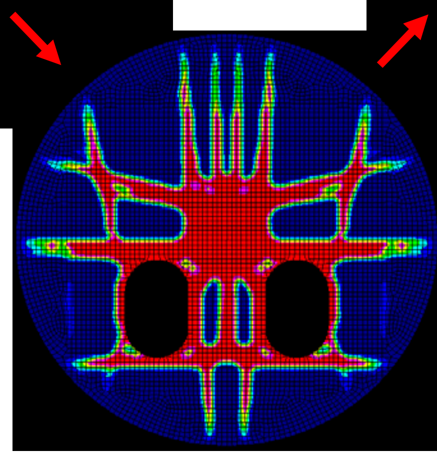
- Optimization between sheet thickness and rib hight

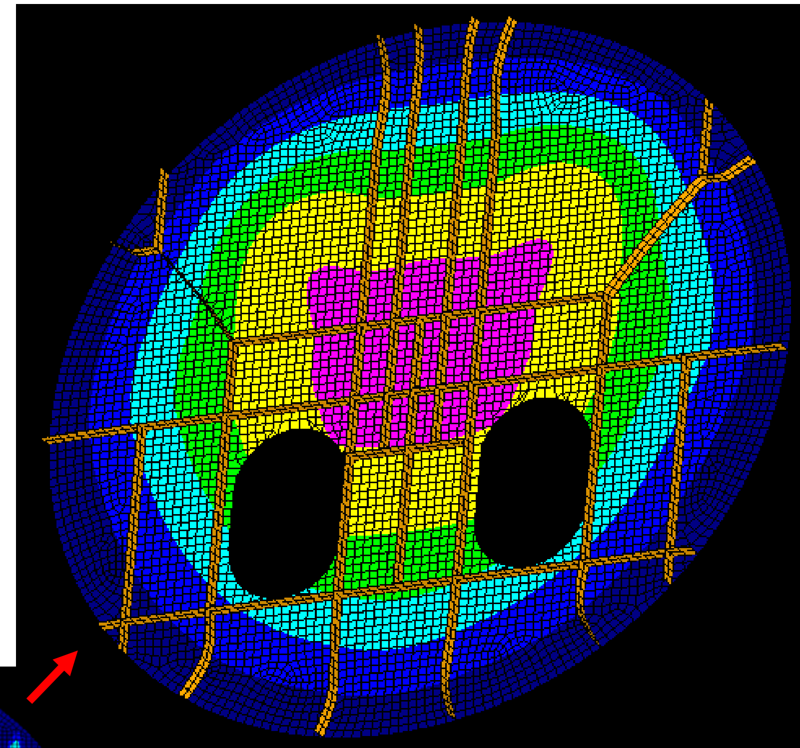
Topology Optimization Example: Bulkhead Stiffeners



Original layout

Max. Deflection:
100%





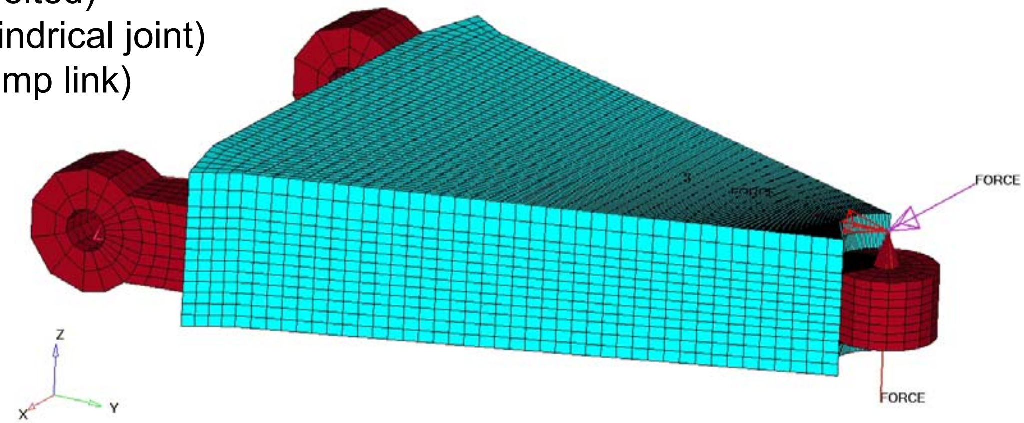
Optimized design

Max. Deflection:
85%

Exercise 4.2: Topologic Optimization of a Control Arm

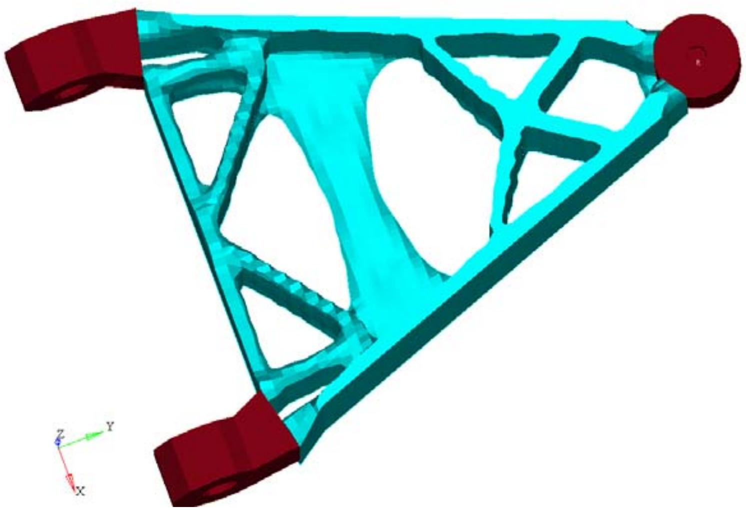
The purpose of this exercise is to determine the best topology or the minimum mass for a control arm that is manufactured using a single draw mode. The arm needs to have a symmetric geometry because it will be used on both sides of the vehicle.

NODE(3) ***X***,***Y*** and ***Z*** . (Bolted)
NODE(4) ***Y*** and ***Z*** . (Cylindrical joint)
NODE(7) ***Z***. (Damp link)



Car turning on a intersection:	corner = (0,1000,0) N	Umax (2699) <= 0.02 mm
Car braking:	brake = (1000,0,0) N	Umax (2699) <= 0.05 mm
Car passing in a pothole:	pothole (0,0,1000) N	Umax (2699) <= 0.04 mm

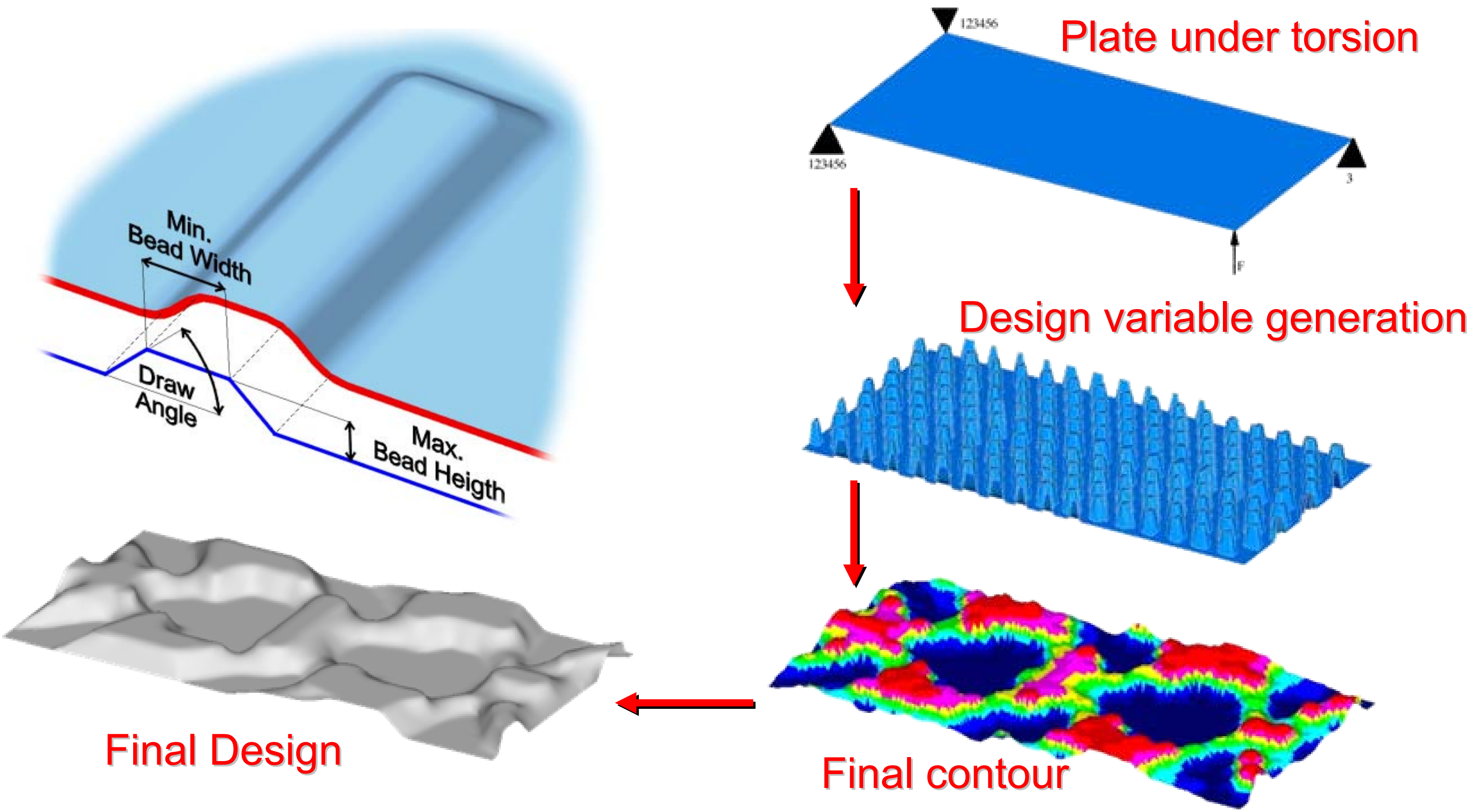
Exercise 4.2: Topologic Optimization of a Control Arm



1. The solution converged to a feasible solution?
2. How much iteration it has take to converge and how much is the final volume of the part?
3. Plot the Iso-contour for the density on the last iteration, does it looks like a manufacturable part?

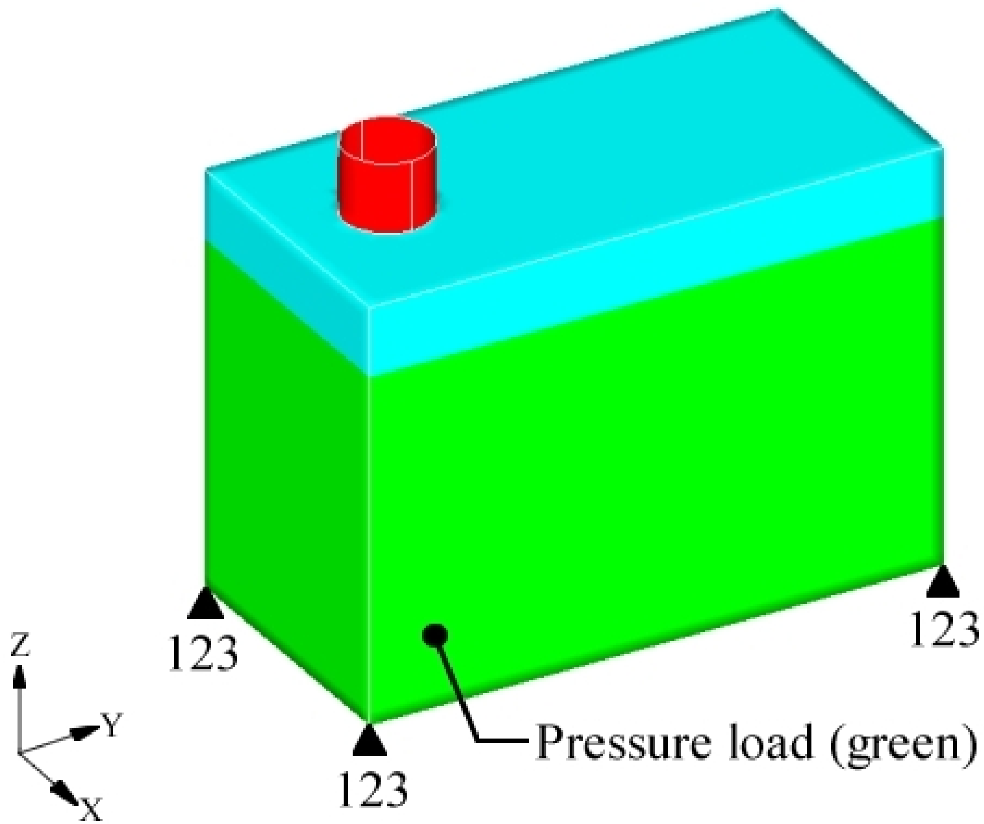
Topography Optimization

- Search for optimal distribution of beads (swages) in shell structures
- Conceptual design method

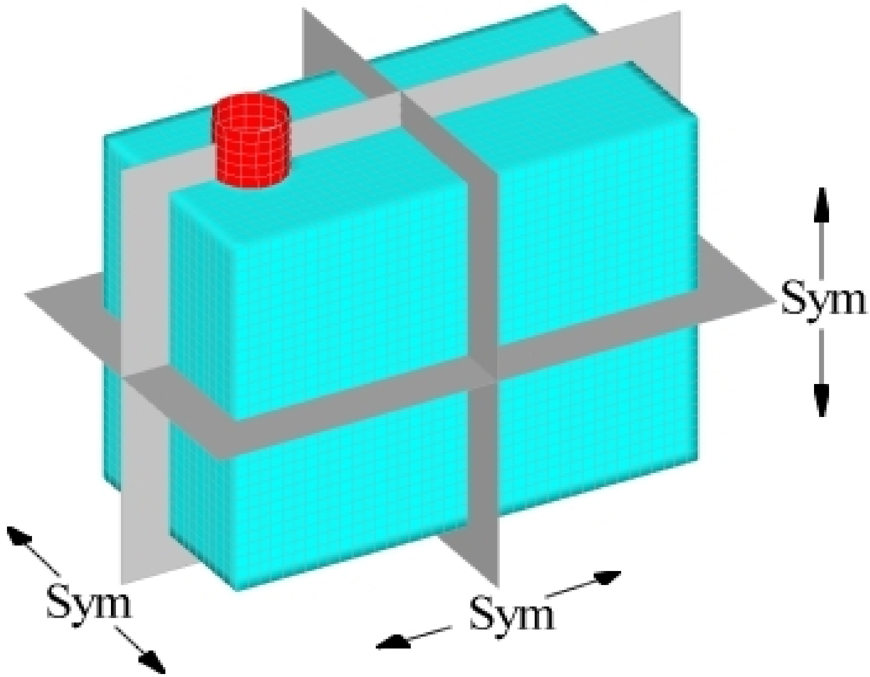


Molded Pressure Tank

- Thin walled tank filled with fluid to be optimized for stiffness



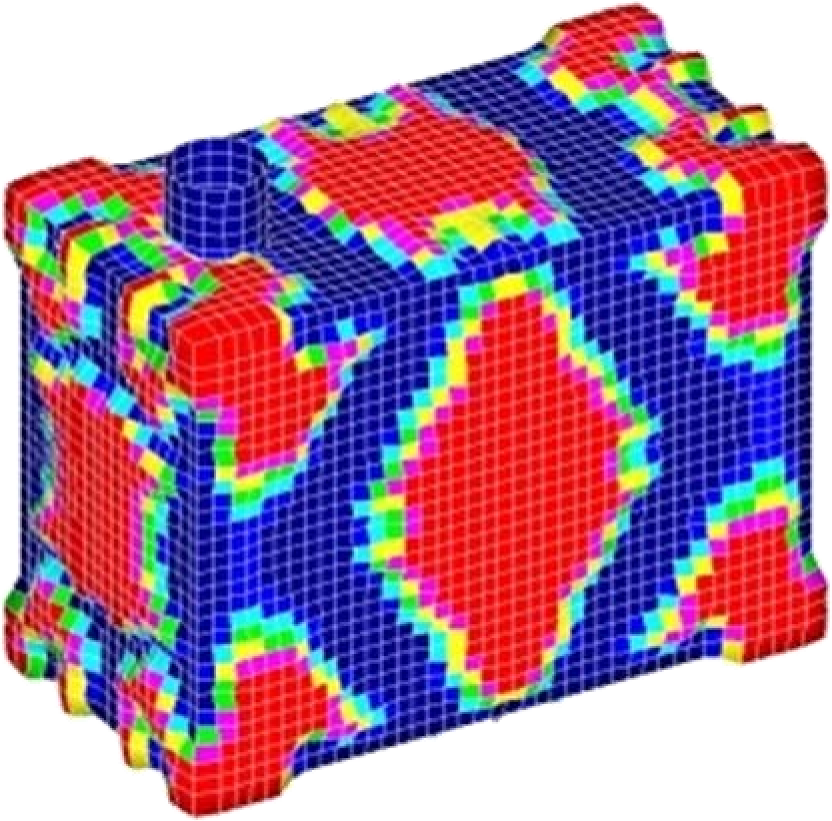
Symmetry



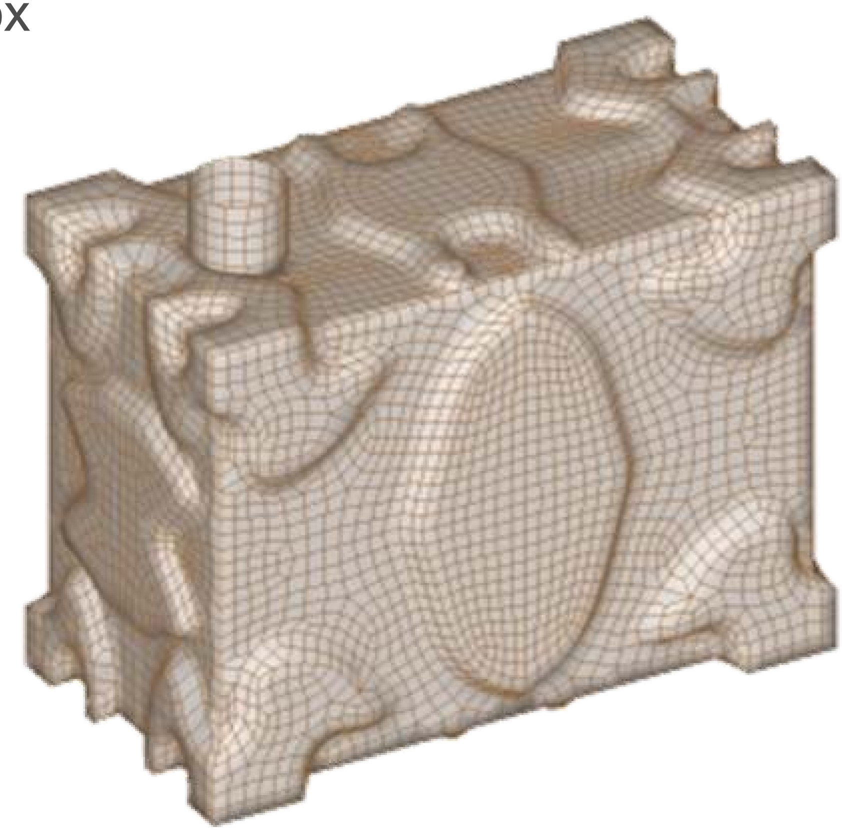
Three orthogonal planes of symmetry are defined

Results

- Reinforcement pattern for pressure box

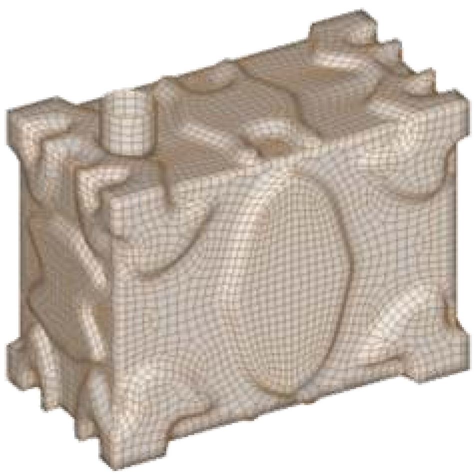


CONTOUR

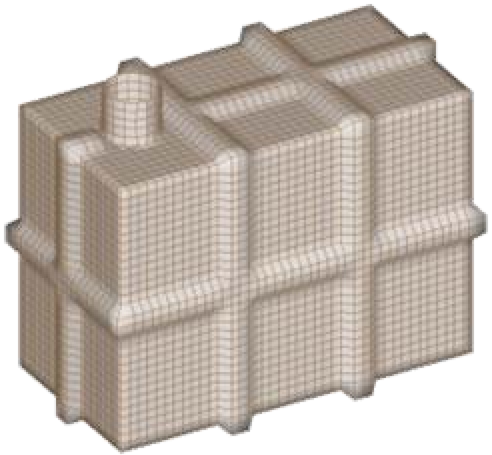


FINAL RESULT

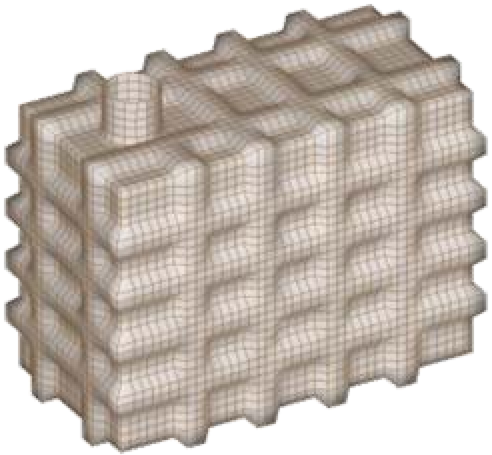
Performance



Max. Deflect: 7.54mm



Max. Deflect: 10.8mm



Max. Deflect: 13.9 mm

- Notice that more ribs doesn’t necessarily mean more stiffness

Topography Optimization

Torsion Plate Example

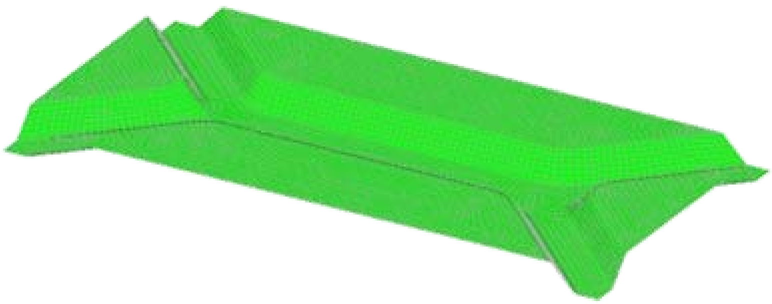


Plate with X-shaped bead pattern

Max. Deflection: 2.23
Max. Stress: 267

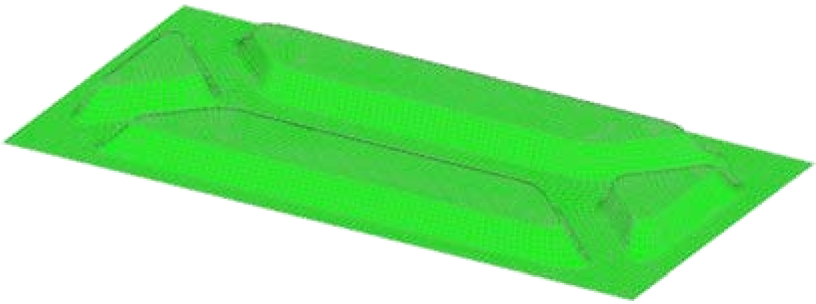
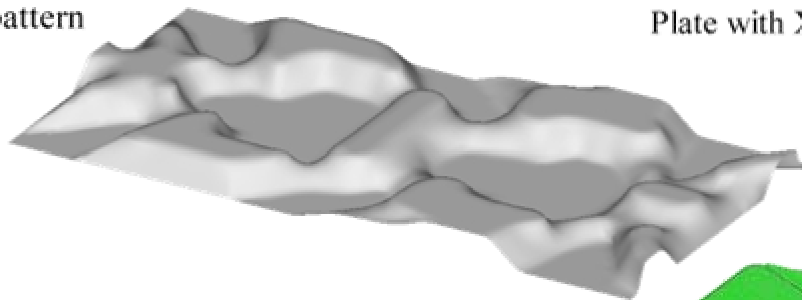


Plate with X-shaped bead pattern and flange

Max. Deflection: 4.41
Max. Stress: 644



Topography Optimization
Max. Deflection: 1.17
Max. Stress: 196

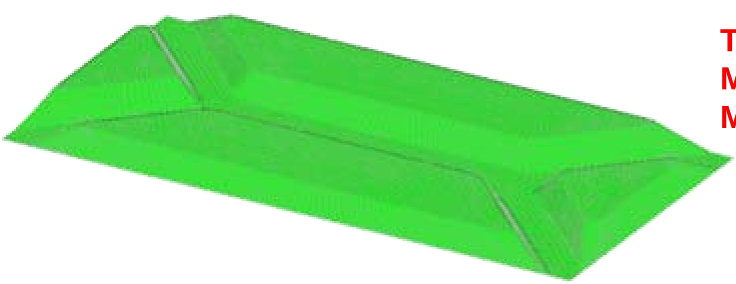


Plate with X-shaped bead pattern and ridged edge

Max. Deflection: 10.57
Max. Stress: 520

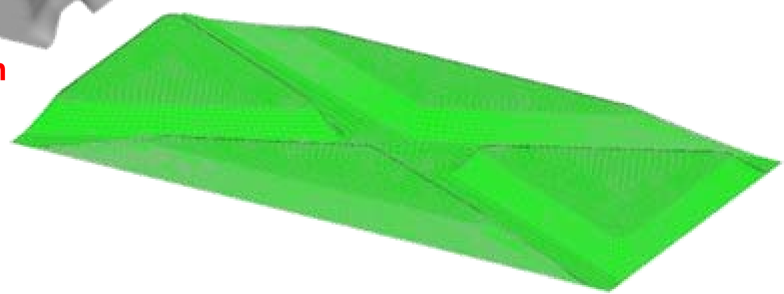


Plate with corner-to-corner bead pattern

Max. Deflection: 6.47
Max. Stress: 434

OptiStruct Input: Topography Optimization

DTPG card – Design Variable definition for topography optimization

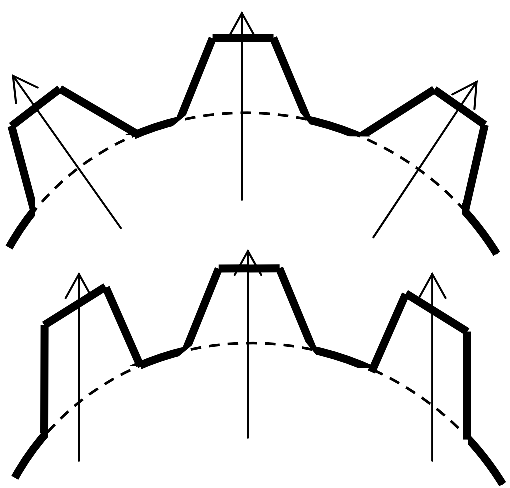
- Definition of Design Variables
  - Nodal movement (shape change) on shell component
  - Each iteration generates new nodal positions
  - Shell, and composite properties (components) can be defined as topography design space.
- Shells
- Composites
- Pattern grouping

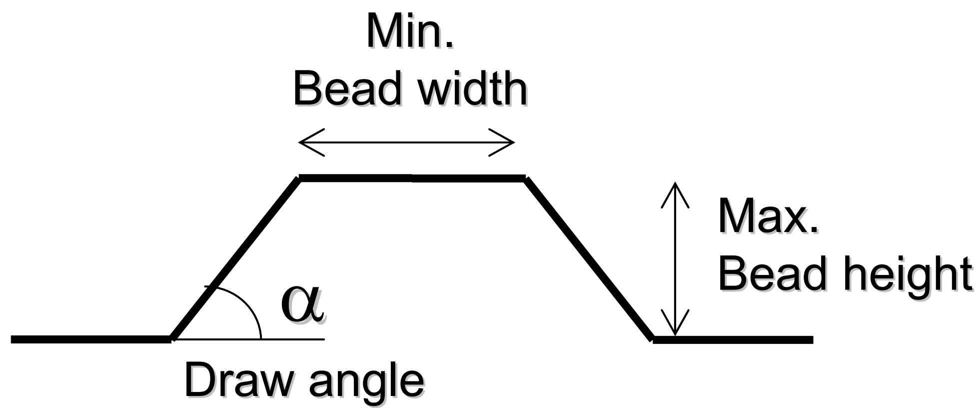
Topography Optimization Setup

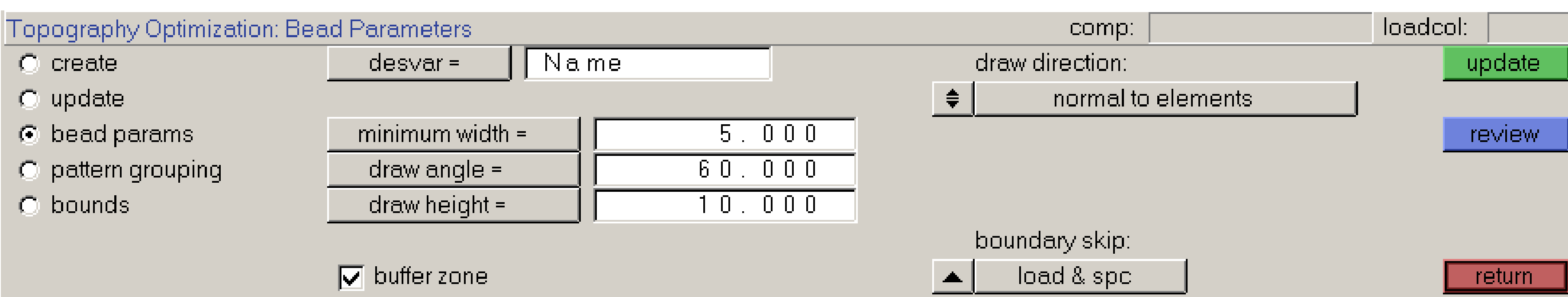
Bead Parameters

normal

global

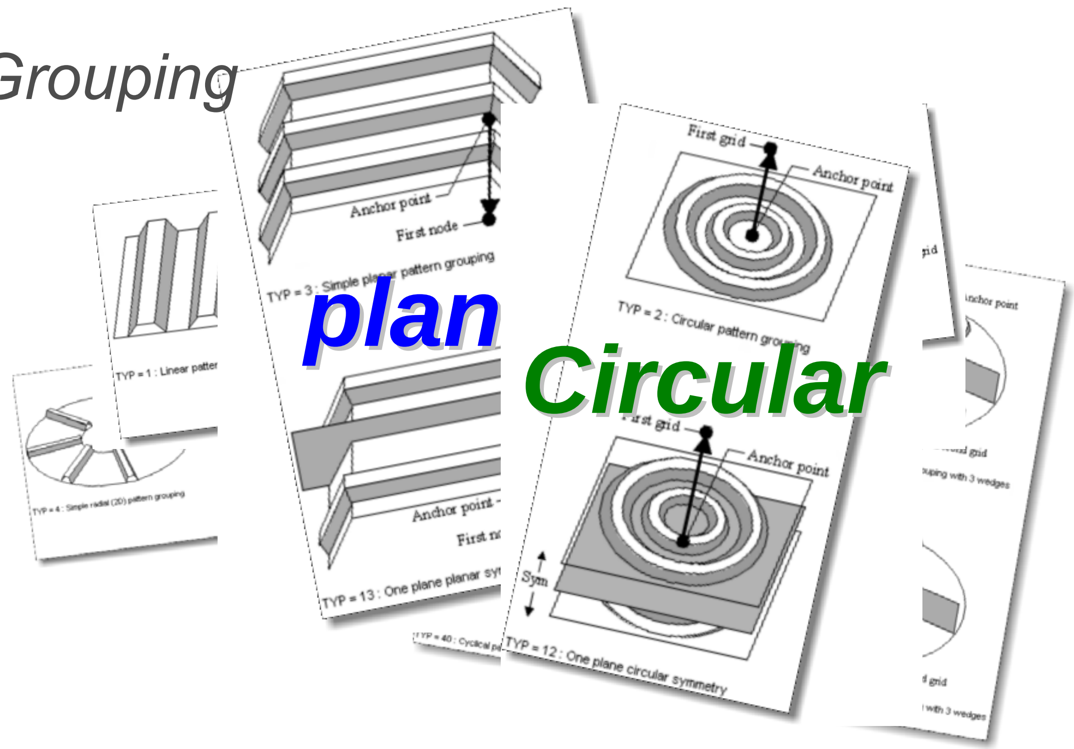






Topography Optimization Setup

Pattern Grouping



select variable grouping pattern

create | desvar = | Name | update

none	radial2d	1-pln sym	cyc 1-pln
linear	cylin	2-plns sym	cyc lin
circular	rad2d+lin	3-plns sym	cyc rad
planar	radial3d	cyclic	cyc lin+rad

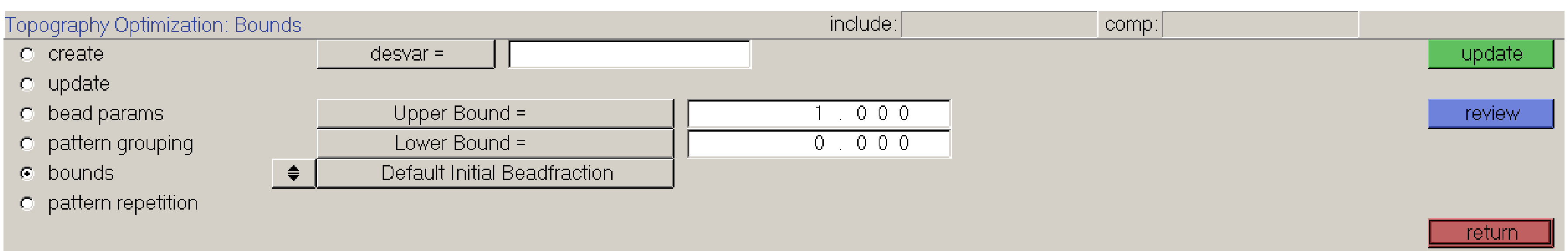
review

return

Topography Optimization Setup

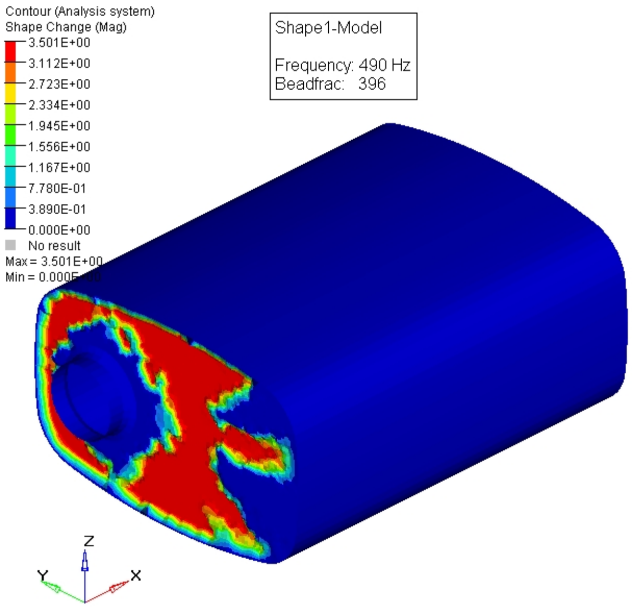
Bounds

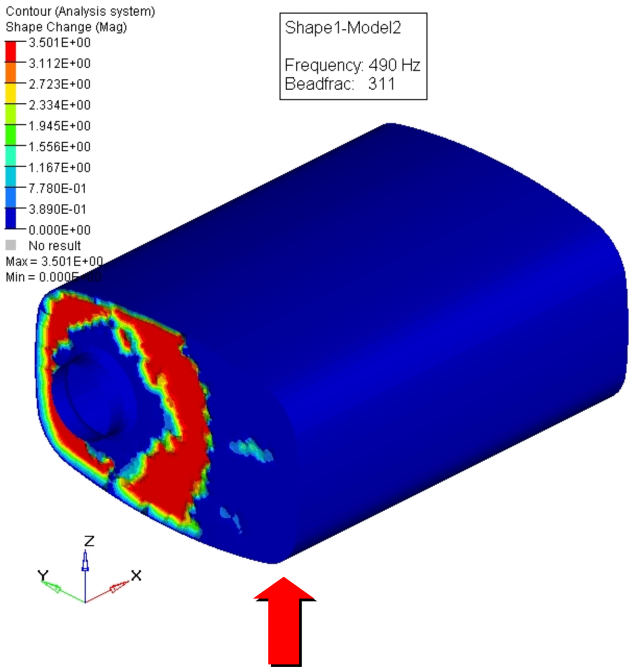
- Beads into one direction
- Beads into two directions
- Initial Bead fraction



Bead discreteness control

- Beadfrac response
  - Used as objective or constraint
  - More discrete results will be achieved with lower beadfrac

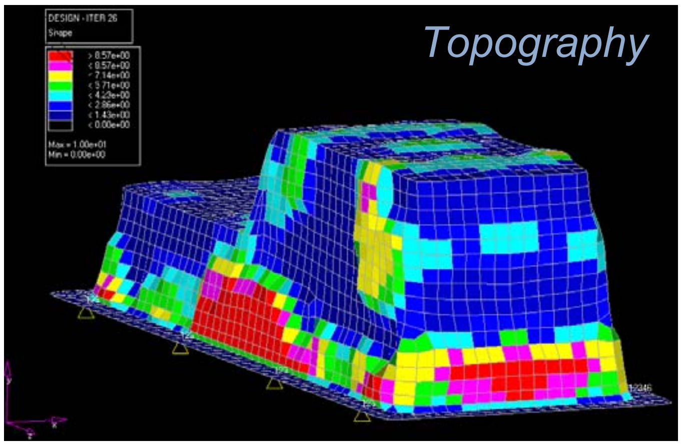


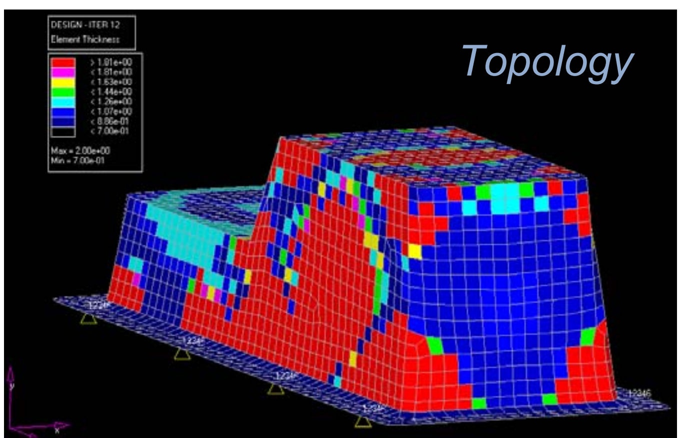


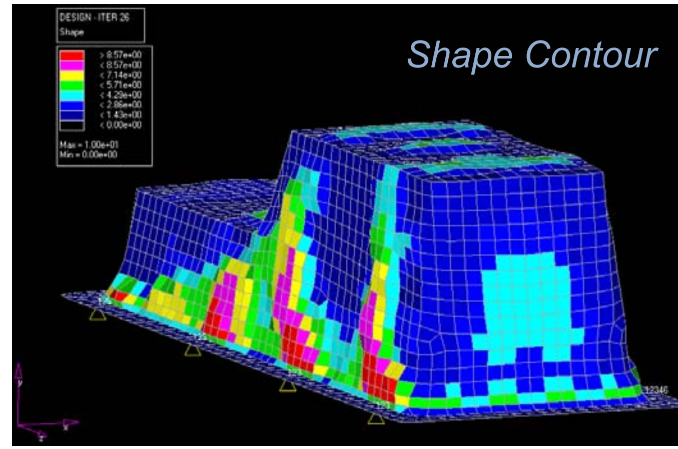
More discrete results

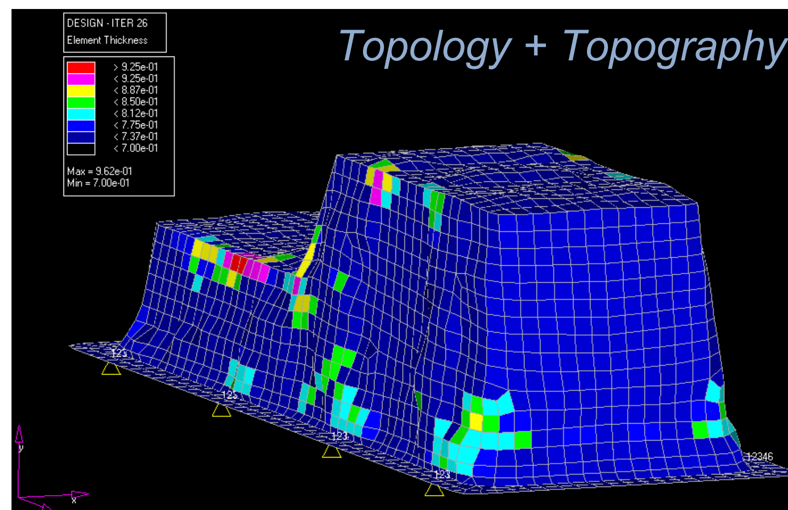
Combining Optimization Types

- Optimization types can be combined
- Example: Topology + Topography



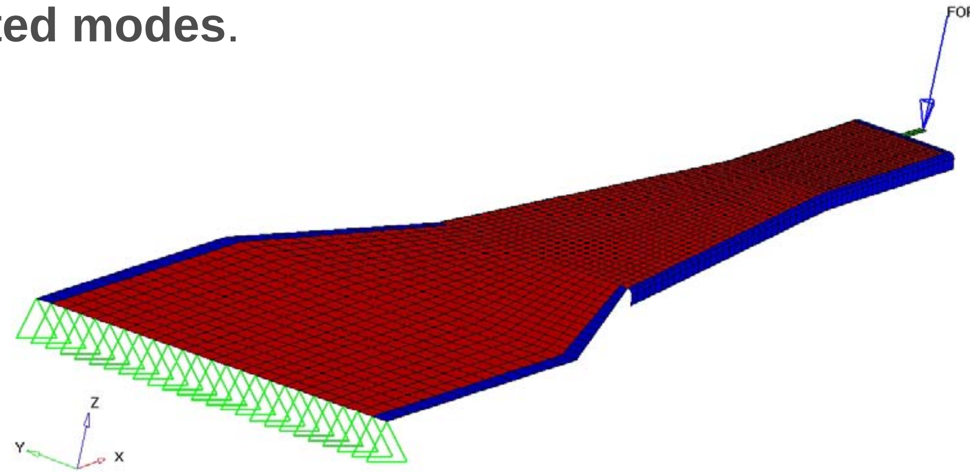






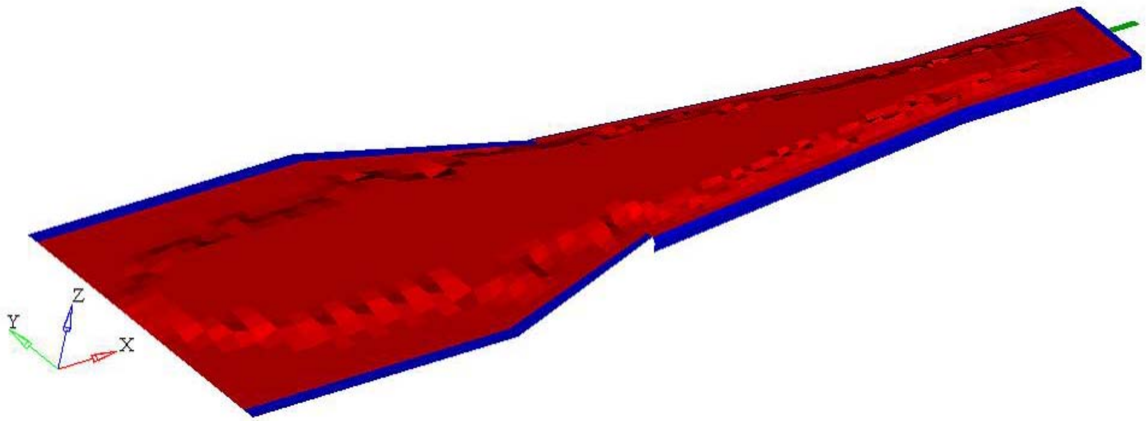
Exercise 4.3: Topography Optimization of a Slider Suspension

On this exercise we will look for the best stamped shape for a slider suspension, the objective function will be a combination of the compliance and the frequency, the objective is to have it as stiffer as possible for the static force, and a stiffer dynamic behavior on the lower frequencies, this function can be defined on OptiStruct as a combined **weighted compliance** and the **weighted modes**.



Objective function:	Minimize the combined **weighted compliance** and the **weighted modes**.
Constraints:	7^{th} Mode > 12 Hz.
Design variables:	**Nodes topography.**

Exercise 4.3: Topography Optimization of a Slider Suspension

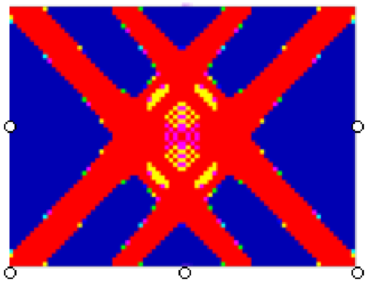


If the student had finish the exercise and wants to try a more advanced setup, these are a small list of things that could improve this result:

1. Add a topologic optimization on the same design space.
2. Add a symmetry plane to the topography and topologic DVs.
3. Increase the Height to 0.2 mm.
4. Use OSSMOOTH to export the geometry.
5. Prepare a HV report to describe the optimization results.
6. Export the final shape and rerun an analysis to check the performance.

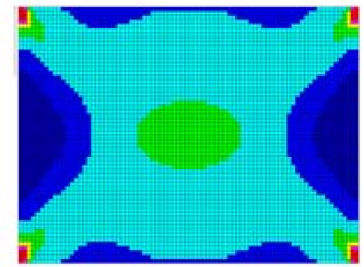
Free-Size Optimization

- Topology optimization



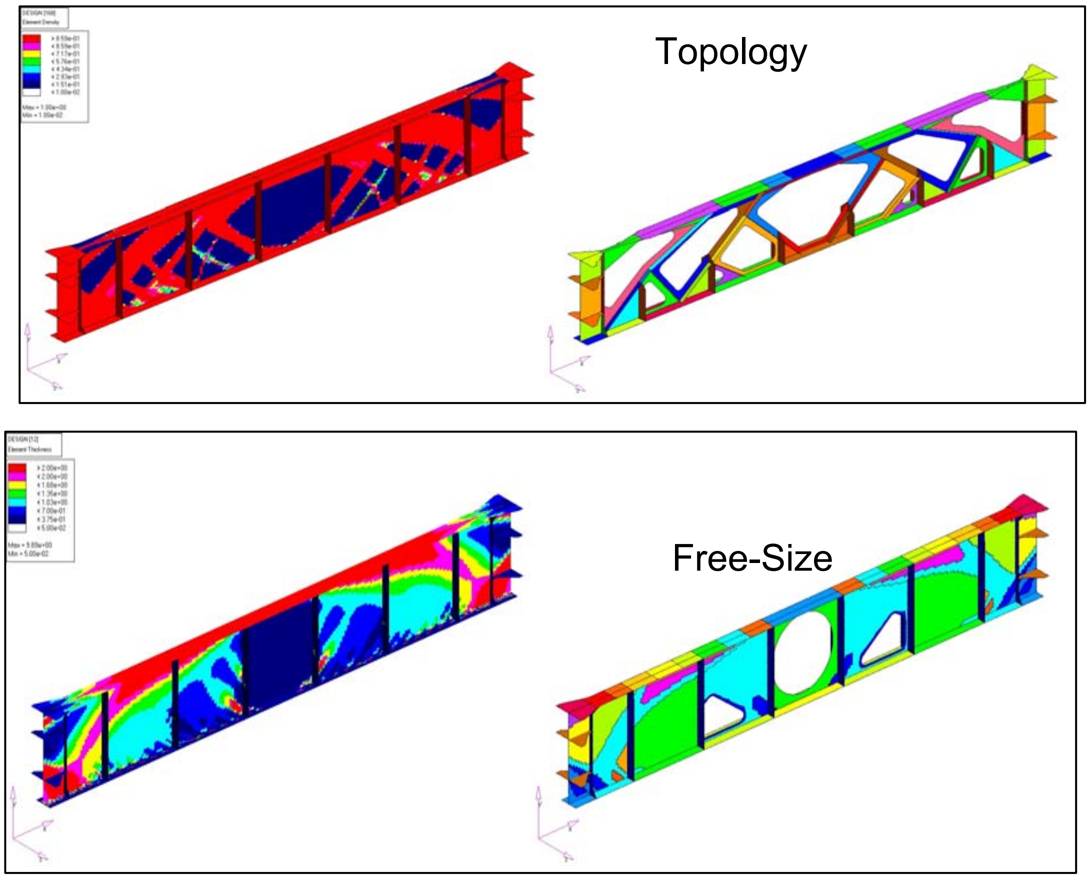
- Design space = Total – Base Thickness
- Design variable – Density
- Poor bending representation of semi-dense elements
- Truss-like design concepts, no shear panels

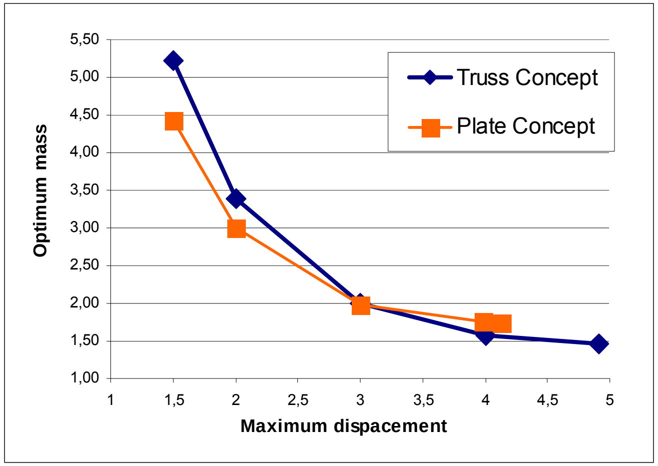
- Free size optimization



- Design variables - Thickness of each element
- Accurate bending representation
- Expandable to composites
- Shear panels possible if they represent the best concept

Free-Size Optimization

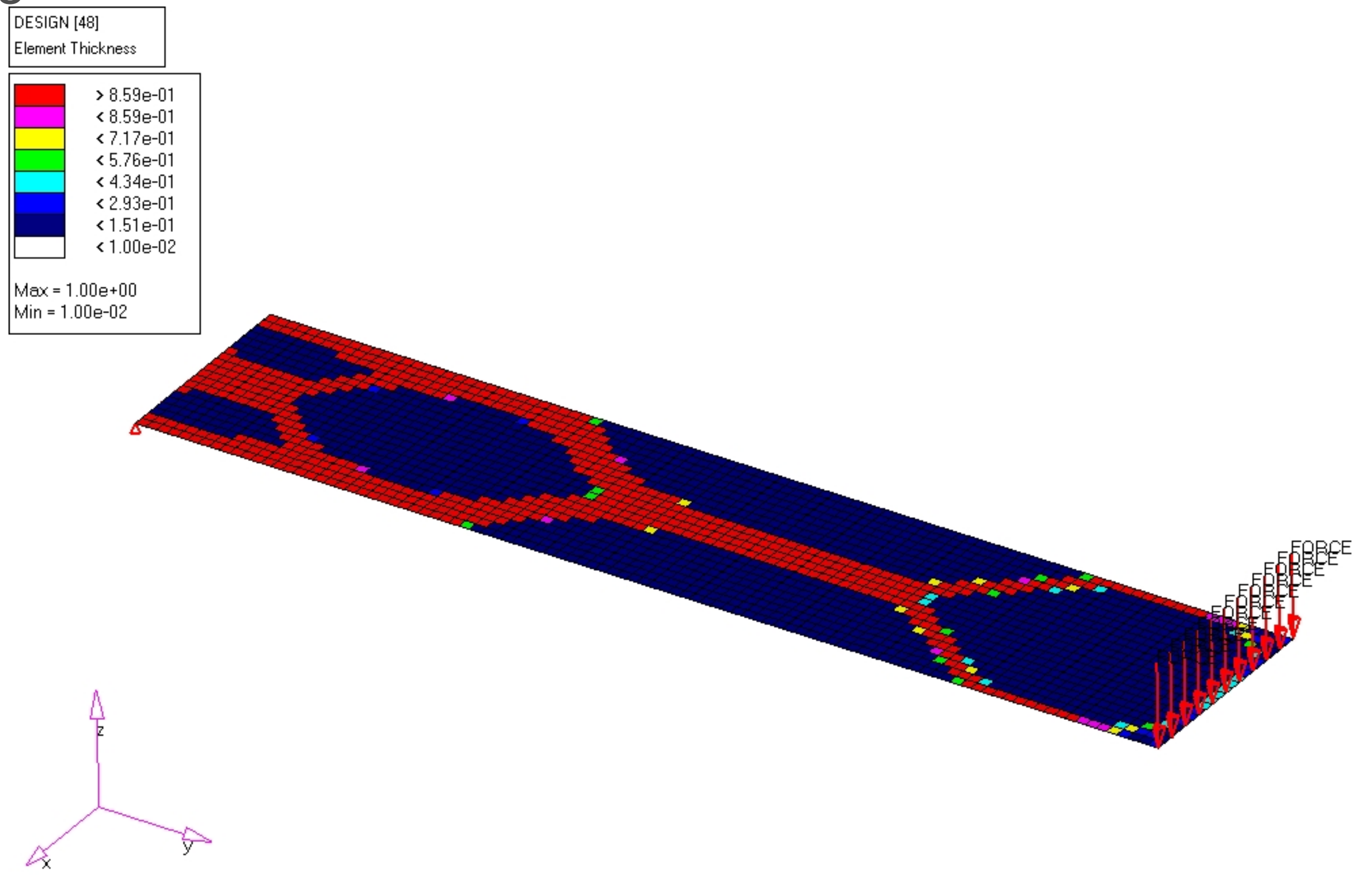




- Concept by topology and Free-Size
- Followed by sizing with buckling and stress constraints in sizing

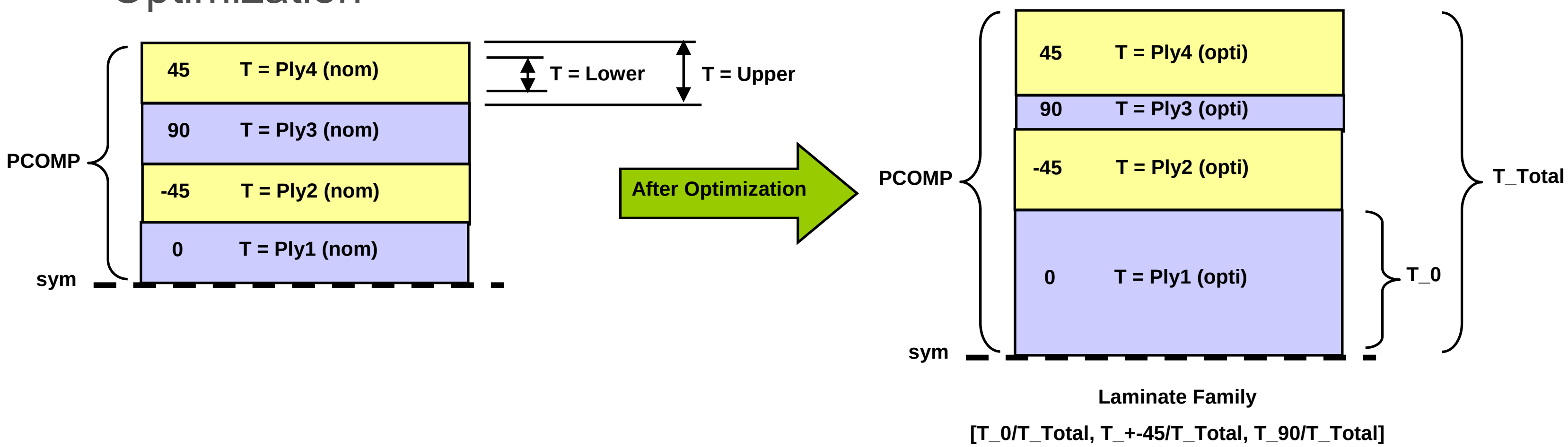
Free-Size Optimization

- The solution will be “discrete” when it needs to be so as the optimum design



Free-Size Optimization on PCOMP

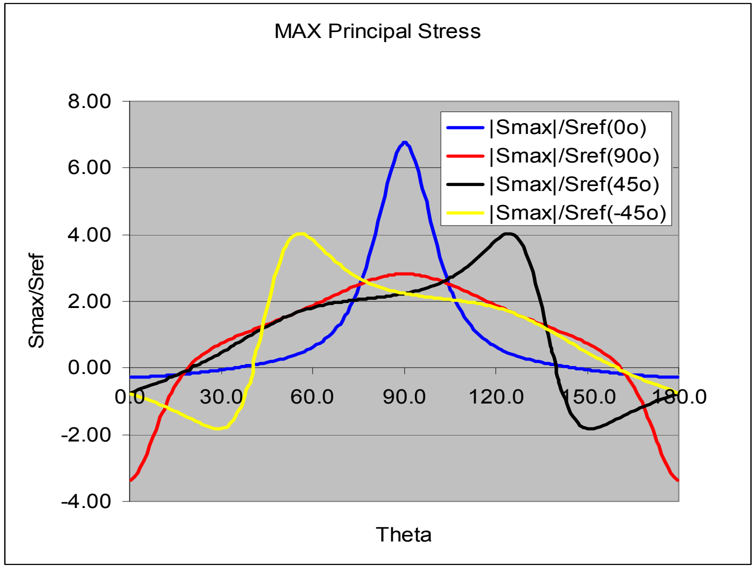
- Composite Free-Size Optimization
  - Each Ply within Each Element has Thickness Design Variable (PCOMP)
  - Stiffness Effected by Laminate Family and Element Thickness in Optimization

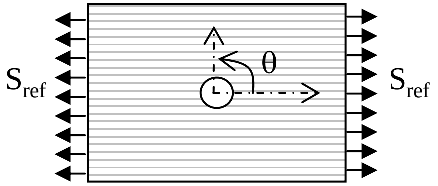


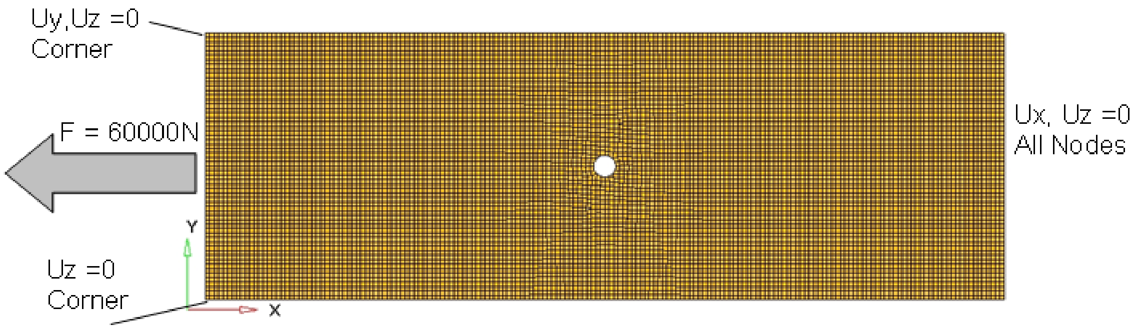
Exercise 4.4: Free-size Optimization of Finite Plate with Hole

The exercise intends to describe the process of setup and post-process of a composite free-size optimization.

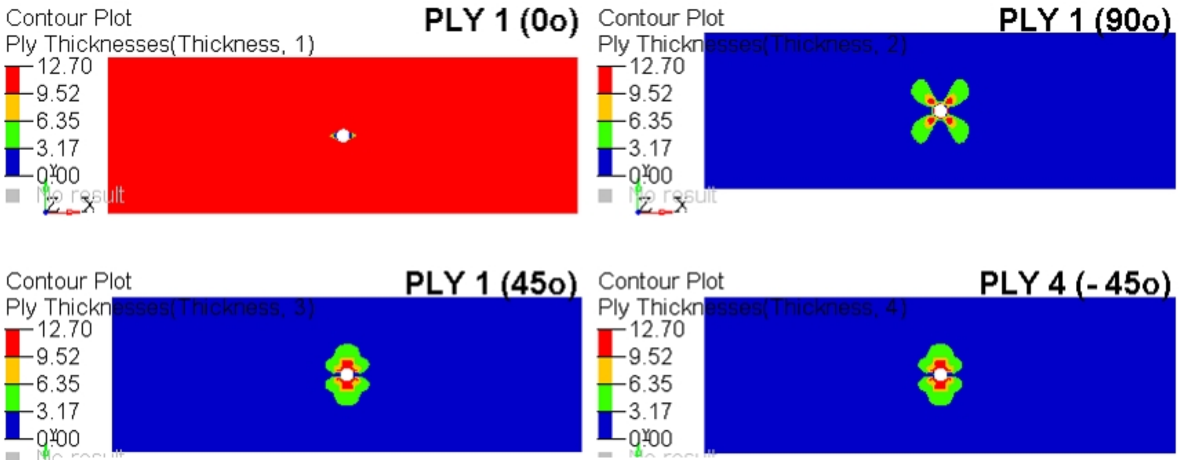
- Objective: minimum weight.
- Configuration: [0, 90, 45, -45] 4 super plies 12.7 mm.
- Constraint: Compliance ≤ 3000 Nmm,
- Manufacturing constraint:
  - Laminate thickness <= 40 mm,
  - 0.5 mm < ply thickness <12.7 mm
  - Balanced 45o and -45o.



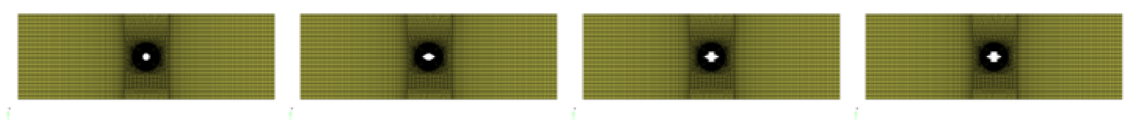




Exercise 4.4: Free-size Optimization of Finite Plate with Hole



0º plies shape



90º plies shape

45º and -45º plies shape

Chapter 5:Fine Tuning Design

Size Optimization

Shape Optimization

Free-shape Optimization

Size Optimization

- Properties are easily sizable
  - Shell Thickness, Beam Sections
  - Masses, Spring Stiffness
- Element (Shells and Beams) properties are a function of design variables
- Gauge Optimization
  - Simplified size optimization
  - Shell thickness ***t* = DV**
  - Gauge panel in HyperMesh – easy setup of thickness optimization for many components

$$p = C_0 + \sum_j C_j DV_j \text{ or}$$

$$p = f(DV_j, C_j)$$

p - Element property

C_j - Constant

Size Optimization

- DESVAR
  - Design variable
- DVPREL1
  - Simple Design Variable to Property RELationship
  - Element property is linear combination of design variables
- DVPREL2
  - User-defined function Design Variable to Property RELationship
  - Defines properties as function of design variables, and table entries

Example: Moment of Inertia for a rectangular beam

$$I(b,h) = \frac{bh^3}{12}$$

Where b and h are beam dimensions

Size Optimization

Example :

Define Shell Thickness of Component ID 5 as a Size Variable.

Initial Thickness: 1.0mm

Thickness Range: 1.0 - 2.5

	PID	MID1	T	MID2	12I/T3	MID3	TS/T	NSM
PSHELL	5	204	1.0					

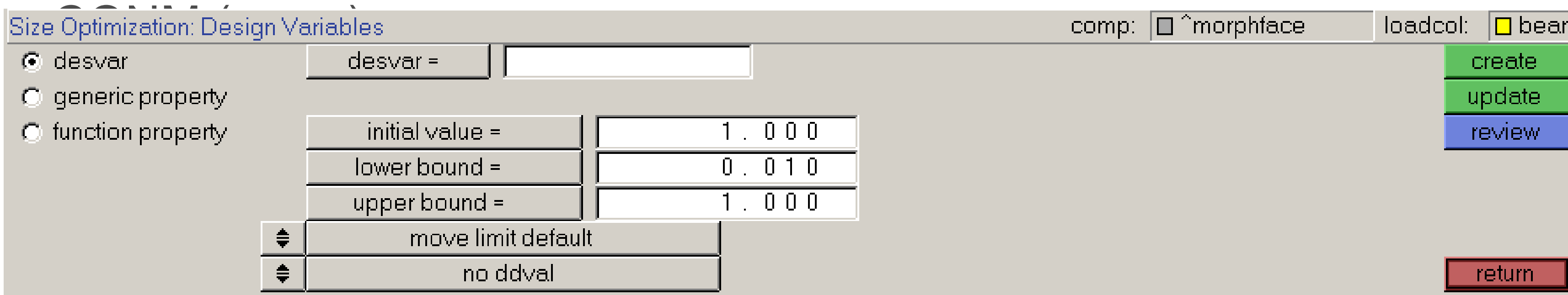
		ID	LABEL	XINIT	XLB	XUB	DELXV	
DESVAR		1	DV001	1.0	1.0	2.5		

		ID	TYPE	PID	FID			C0
DVPREL1		88	PSHELL	5	4			0.0
		DVID1	COEF1	DVID2	COEF2	Etc		
		1	1					

Size Optimization

Definition of Design Variables

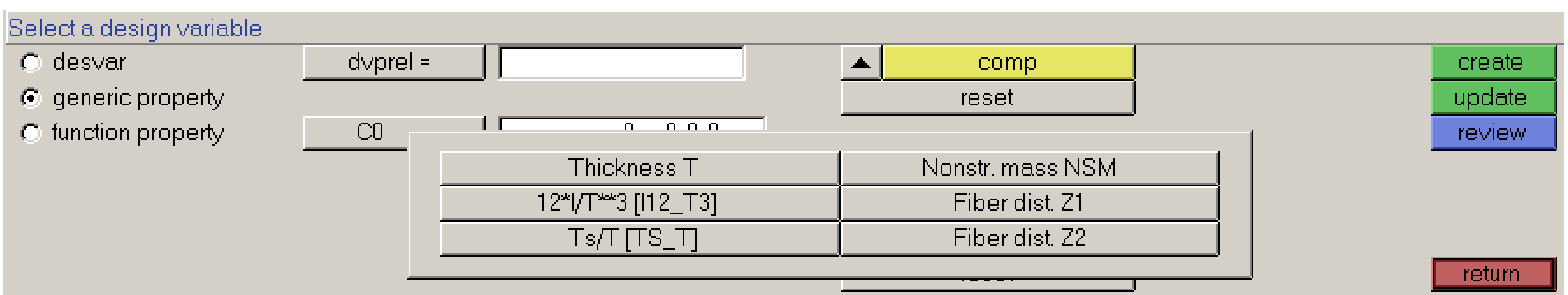
- Definition of initial value, lower bound, upper bound
- PROD (area)
- PBAR,PBEAM (Area, Moment of Inertia, etc.)
- PBARL,PBEAML (height, width, etc.)
- PELAS (stiffness)



Size Optimization

Build relationship between design variables and properties

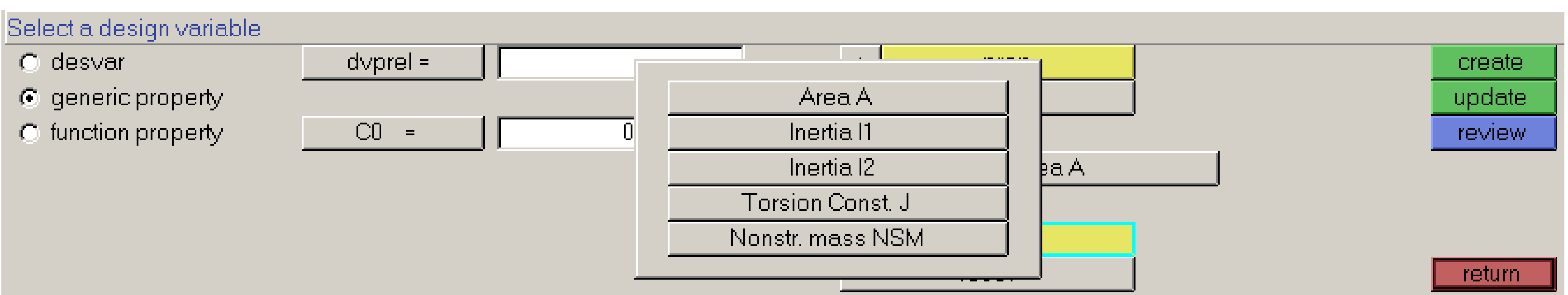
- PROD (area)
- PBAR,PBEAM (Area, Moment of Inertia, etc.)
- PBARL,PBEAML (height, width, etc.)
- PELAS (stiffness)
- CONM2 (mass)



Size Optimization

Build relationship between design variables and properties using functions

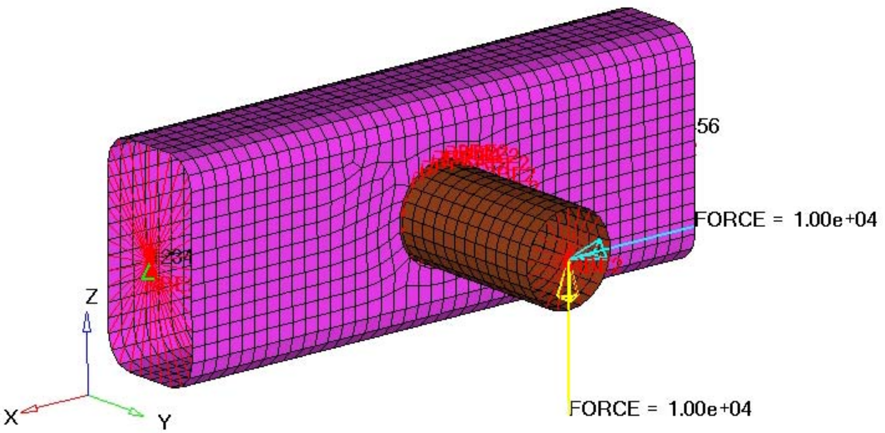
- A = f (b,h) = b*h
- I1 = f (b,h) = 1/12*b*h^3
- I2 = f (b,h) = 1/12*b^3*h
- J = f (b,h) = ...



Exercise 5.1 – Size Optimization of a Rail Joint

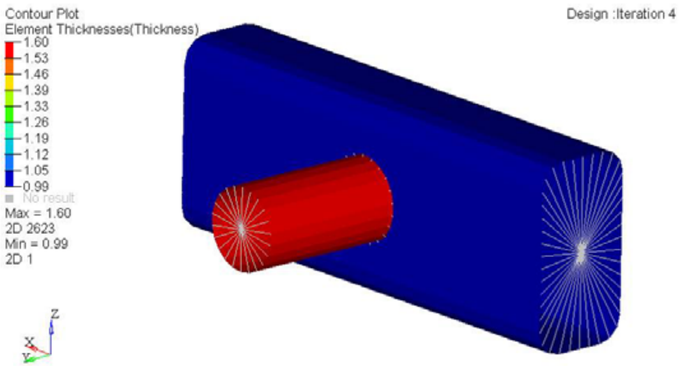
This exercise demonstrates how to perform a size optimization on an automobile rail joint modeled with shell elements.

- The structural model with loads and constraints applied is shown in the figure.
- The deflection at the end of the tubular cross-member should be limited.
- The optimal solution would use as little material as possible.



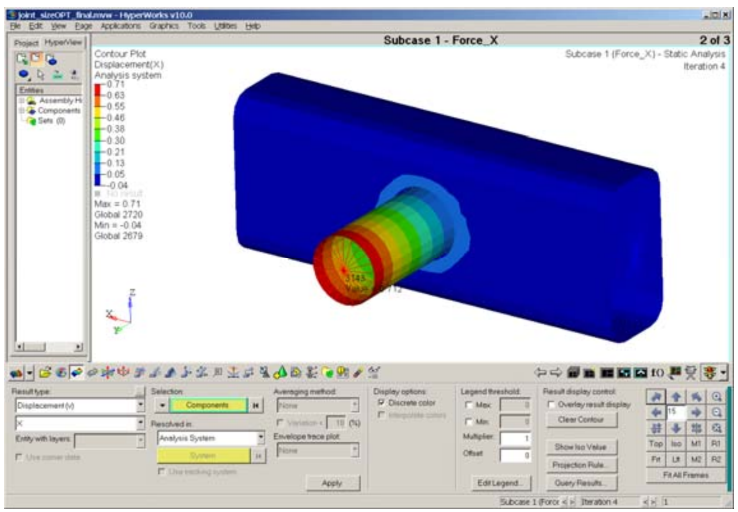
Objective:	Minimize volume.
Constraints:	U_x (max) ≤ 0.9. U_z(max) ≤ 1.6.
Design variables:	Gauges of the two parts.

Exercise 5.1 – Size Optimization of a Rail Joint

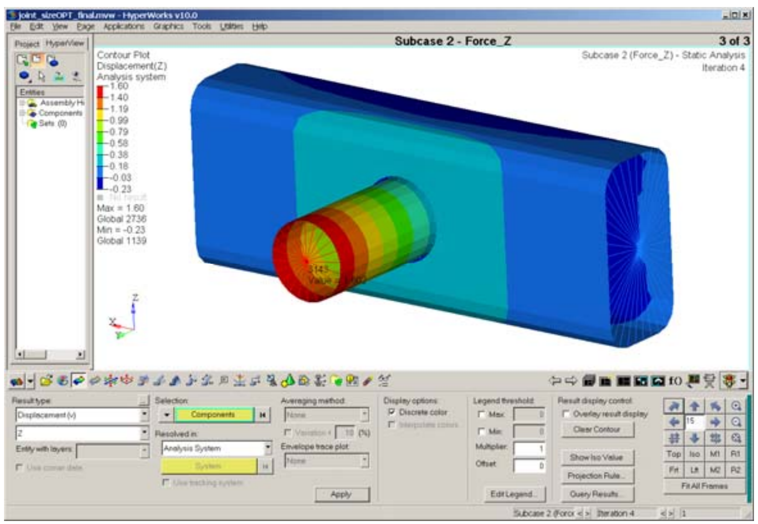


Thickness

1. The solution converged to a feasible solution?
2. How much iteration it has take to converge and how much is the final volume of the part?
3. What are the resulting gauges for the rail and tube?



Ux (max) ≤ 0.9.

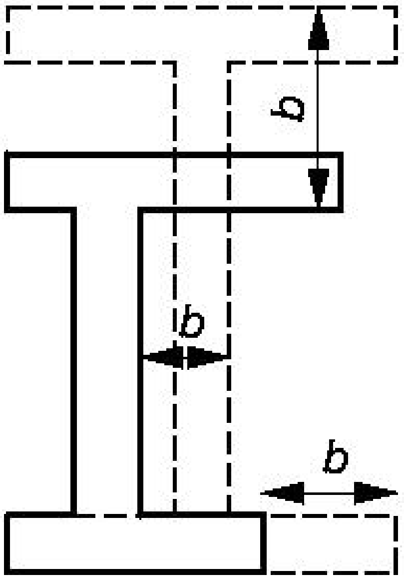


Uz(max) ≤ 1.6.

Shape Optimization

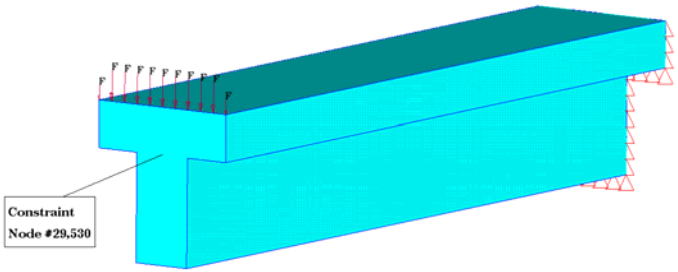
Modify geometry to achieve objective

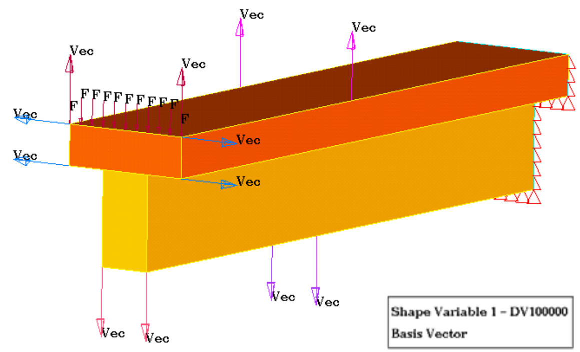
- Fillet Radii
- Rib Height
- Channel Depth / Width
- Solid Cross Sections



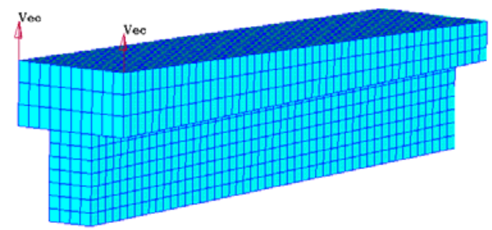
(b)

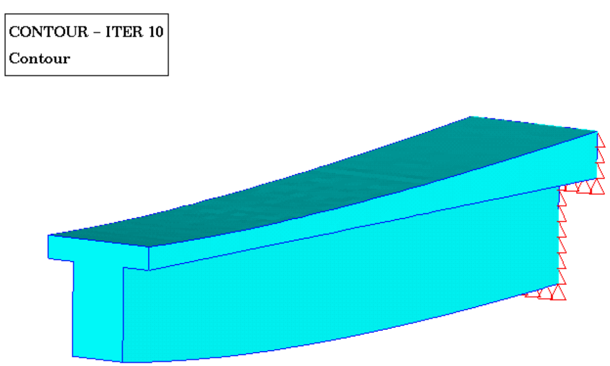
Shape Optimization

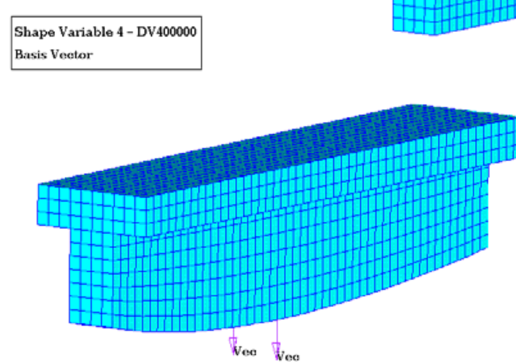




Initial design



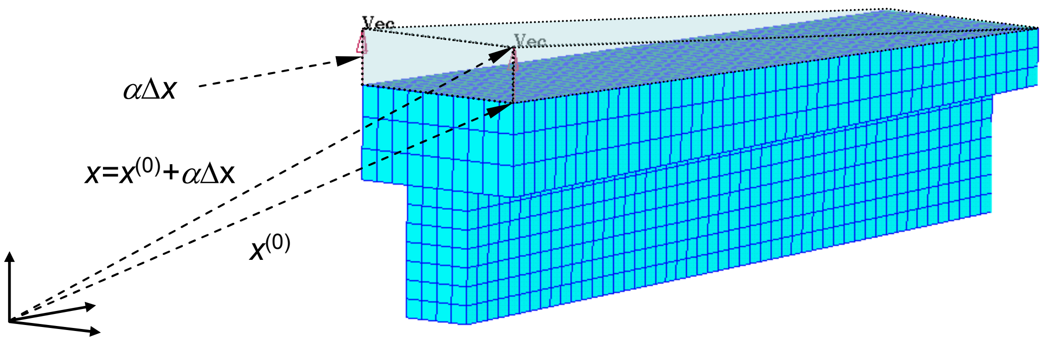




Final design

Shape Optimization

Single nodal movement due to single shape variable



Original location: $X^{(0)} = \{x_1^{(0)}, x_2^{(0)} x_3^{(0)}, \cdots, x_n^{(0)}\}$

Perturbations (DVGRID): $\Delta X = \{\Delta x_1, \Delta x_2, \Delta x_3, \cdots, \Delta x_n\}$

Magnitude of perturbations (DESVAR): $\alpha = \{\alpha_1, \alpha_2, \alpha_3, \cdots, \alpha_n\}$

Mesh nodal movement:

$$X = X^{(0)} + \sum_{j=1}^{n} \alpha_j \Delta X_j$$

Shape Optimization

- DESVAR
  - Design variable
  - Card Image

	ID	LABEL	XINIT	XLB	XUB	DELXV
DESVAR	1	DV001	0.0	-1.0	1.0	

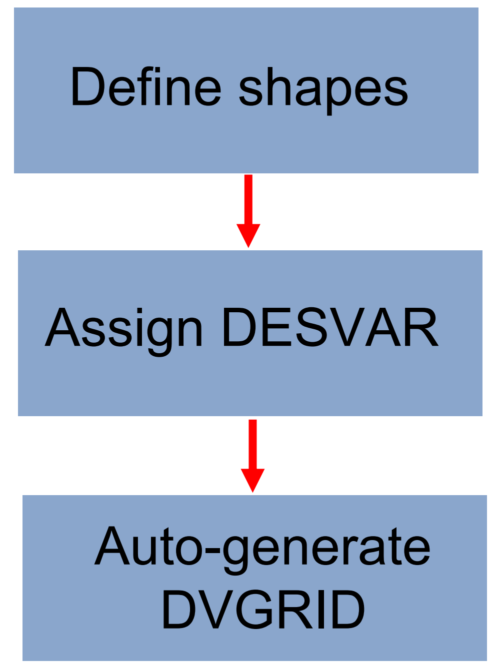
- DVGRID
  - Unit mesh perturbations
  - Total perturbation due to a single design variable is DESVAR * DVGRID
  - Card Image

	DVID	GID	CID	COEFF	X	Y	Z
DVGRID	1	1032	0	1.0	1.0	0.0	0.0

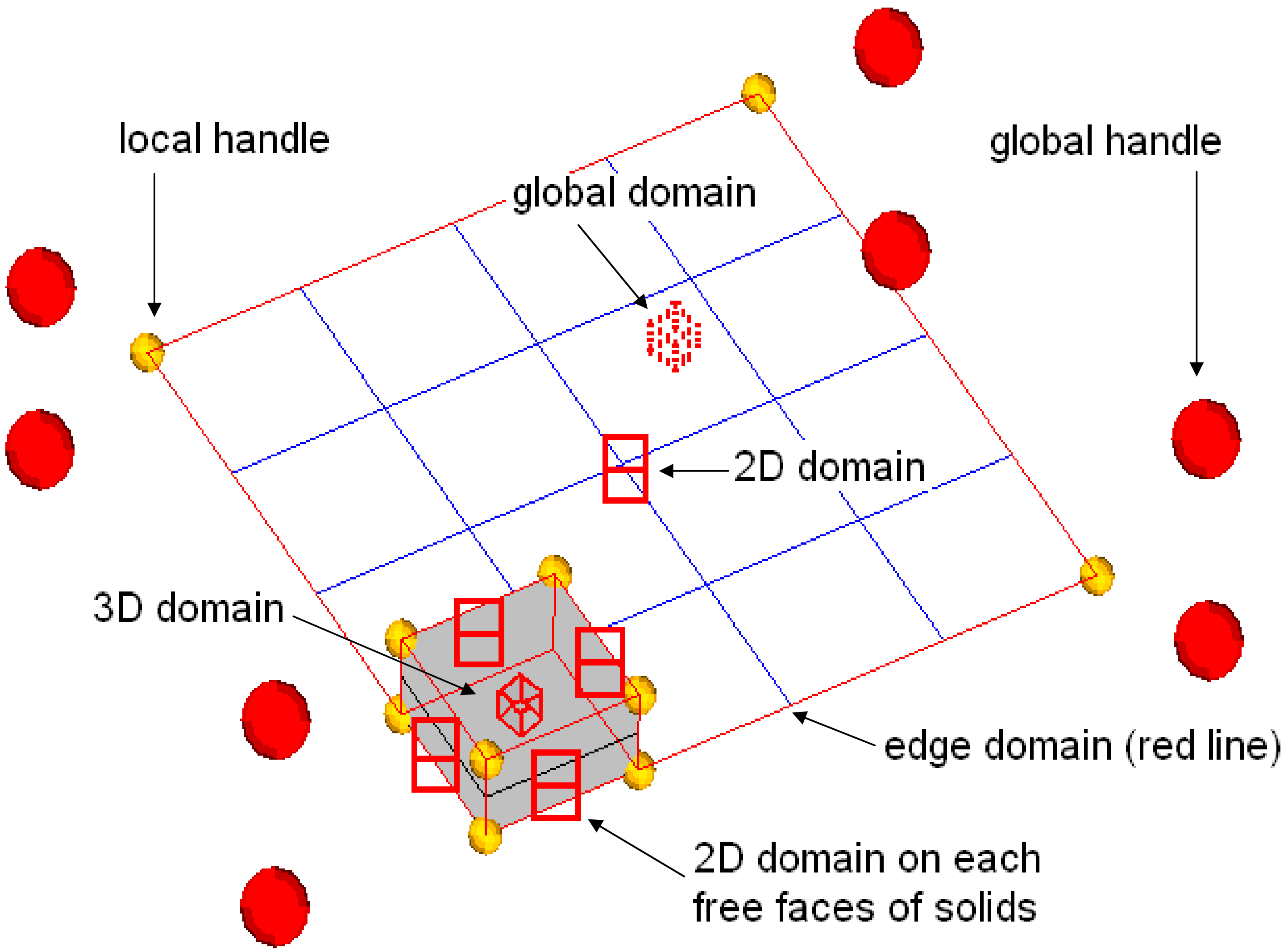
Shape Optimization

Defining Shapes in HyperMesh

- Shapes need to be defined first
  - Mesh morphing (HyperMorph)
  - Perturbations
- Mesh topology must be maintained
- Shapes are then assigned to design variables
- Perturbations are exported with the OptiStruct input deck



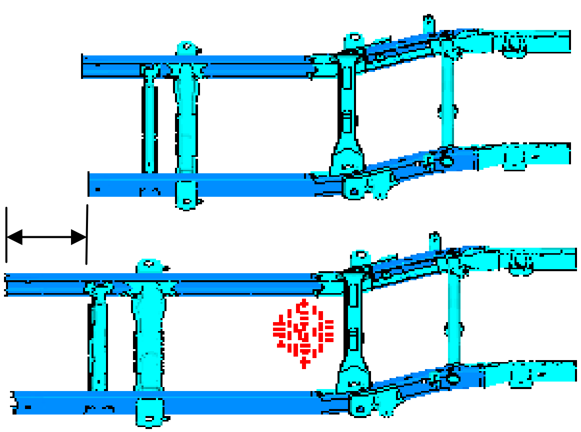
Morphing with Domain and Handle



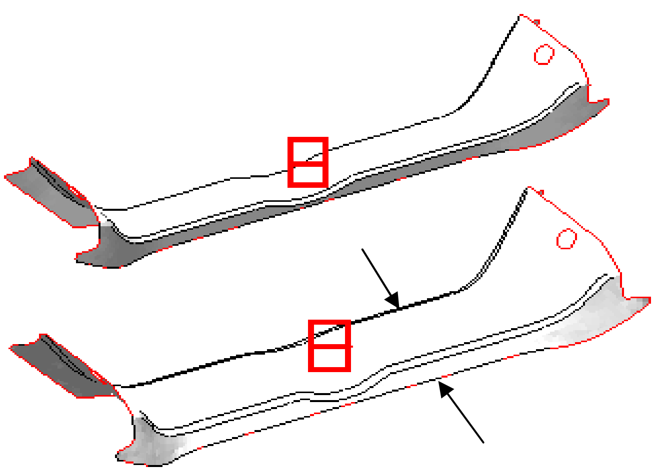
Morphing with Domain and Handle

- Domain: a grouping of elements and nodes that are influenced together during morphing
- Global domain: a single domain which can influence every node in the model.
- Local domains: include1D domain, 2D domain, 3D domain and edge domain. A model can have multiple local domains for morphing different local areas.

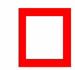
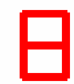
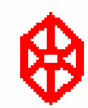
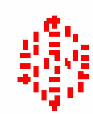
example of global domain



example of local domain



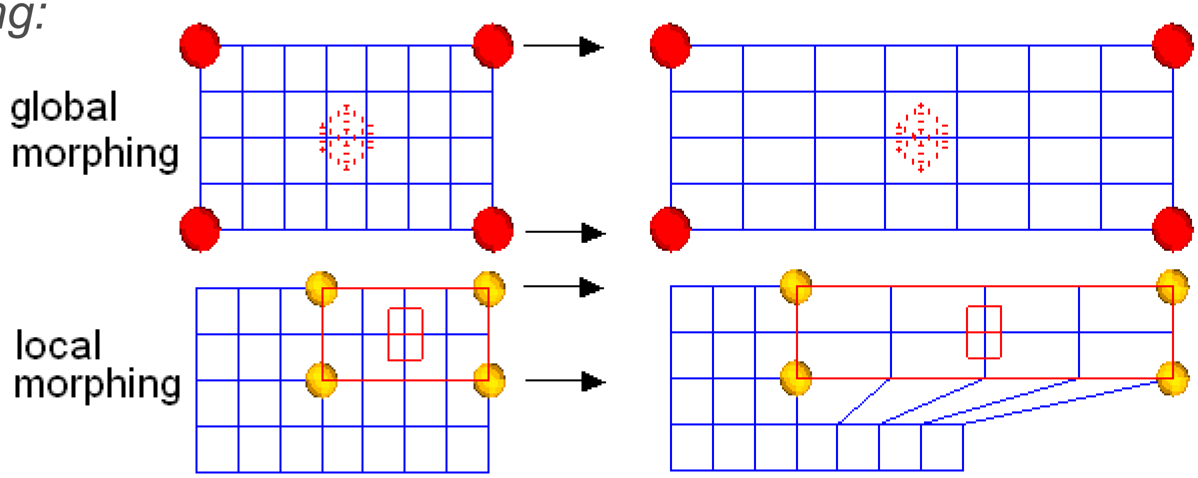
Morphing with Domain and Handle

Domain type	Content	Symbol in HM
1D domains	Contain a group of 1D elements such as bars and rigid elements.	
2D domains	Contain a group of shell elements	
3D domains	Contain a group of solid elements.	
edge domains	Contain a series of nodes and are commonly found along the edges of 2D and 3D domains.	Red lines around the edges of all 2D domains
global domain	Consists of the entire model.	

Morphing with Domain and Handle

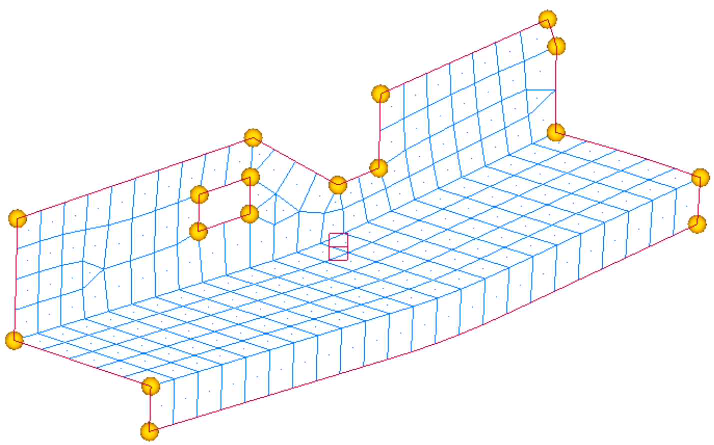
- Handle: control point used to alter the shape of a domain
- Handle Influence: describes how the movement of a handle relates to the nodes in the domain
- Global Handle: Handles affecting the global domain. Movement of a global handle affects every node within a model, allowing large scale shape changes
- Local Handle: Handle affecting local domains. Local handles can only influence the nodes contained within the domains they are associated with

Types of Morphing:

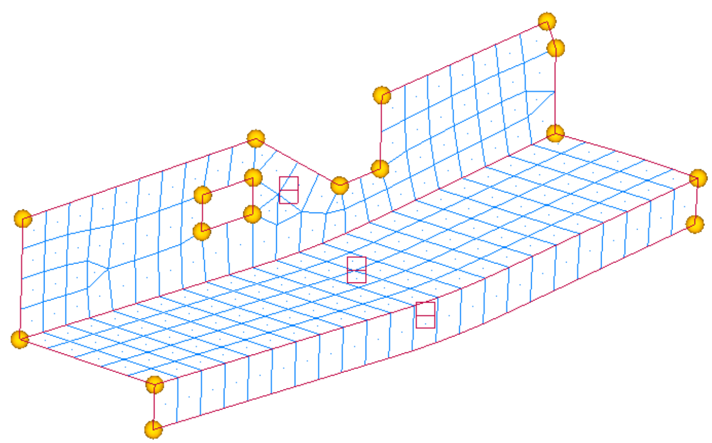


Morphing with Domain and Handle

- Partitioning: Division of a 2-D morphing domain into smaller 2-D domains based on feature angle as specified by the angle and curve tolerance
- Domain Angle: The angle between the normals of 2 adjacent elements. When the value is exceeded, a partition break will be created with an edge between the two elements
- Curve Tolerance: A parameter used to determine if a mesh is curved or planar. Similar to the domain angle, a partition break will be created if the value is exceeded



Without partitioning
partitioning



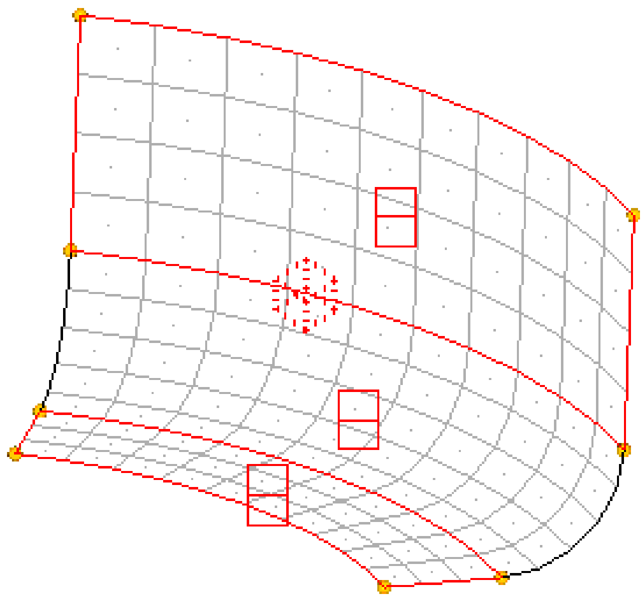
With

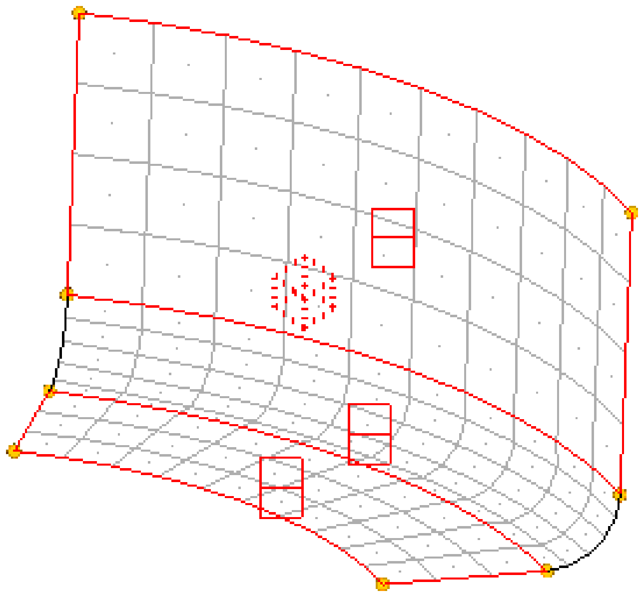
Morphing with Domain and Handle

Alter Dimensions / Radius and Curvature: Change the radius or curvatures of edge domains

Curvature is a scalar applied to the radius for edge domains with varying curvature

Options control changes with respect to curve center, ends or midpoint

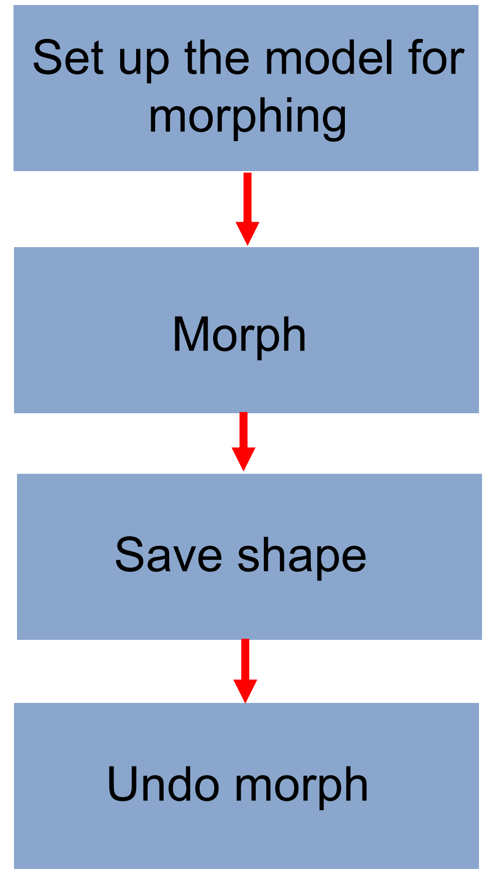




Shape definition for Optimization

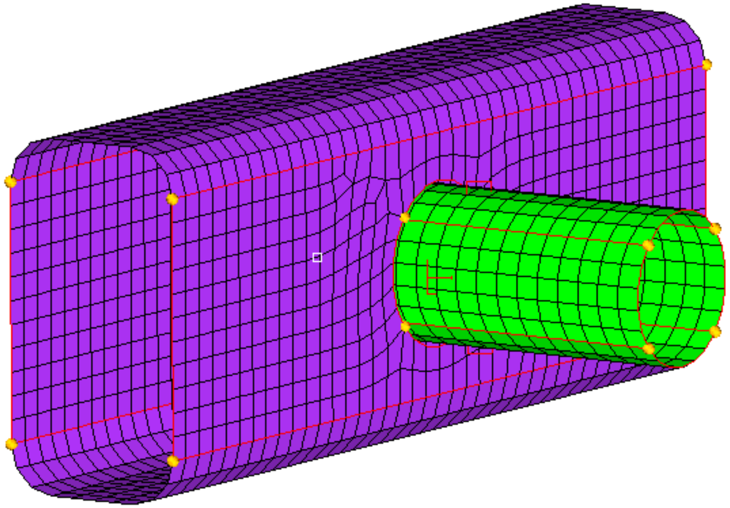
Using HyperMorph

- Use any of the four morphing methods
- Morph the model to the desired shape.
- Save the shape
- Undo the shape
- Save the HyperMesh session file.
- Create a desvar (design variable)
- Run Optimization.



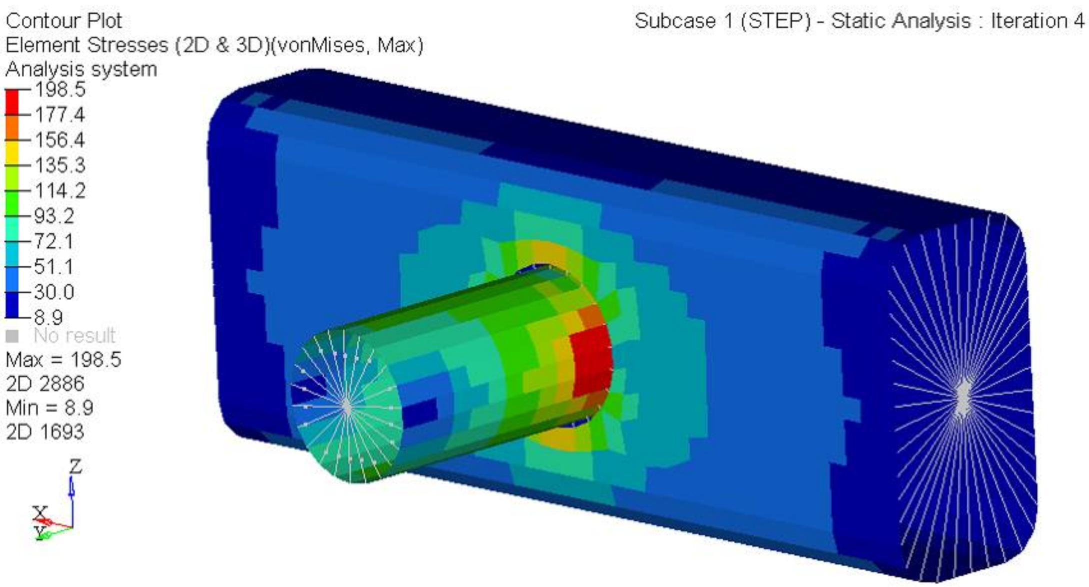
Exercise 5.2 – Shape Optimization of a Rail Joint

- In this exercise you perform a shape optimization on a rail-joint. The rail-joint is made of shell elements and has one load case. The shape of the joint is modified to satisfy stress constraints while minimizing mass.



Objective:	Minimize mass
Constraint:	Maximum von Mises stress of the joint < 200 MPa
Design variables:	Shape variables

Exercise 5.2 – Shape Optimization of a Rail Joint

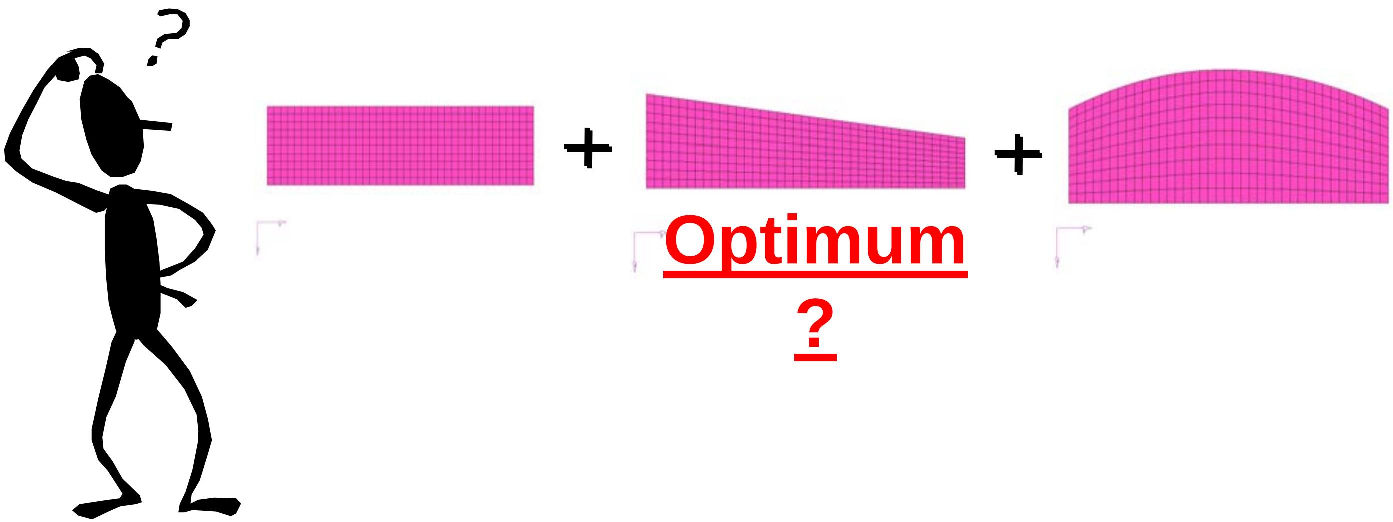


Maximum von Mises stress of the joint < 200 MPa

Is your design objective of minimizing the volume obtained? If not, can you explain why?
Are your design constraints satisfied?
Which shape has the most influence in this problem setup?
What is the percentage decrease in compliance?
Can size optimization be introduced to the joint?

Free Shape Optimization

- No user-defined shape perturbation vector is necessary
  - Reduce the effort to guess what would be the optimum shape
- Free Shape optimization uses a proprietary optimization technique developed by Altair, wherein the outer boundary of a structure is altered to meet with pre-defined objectives and constraints
- Can be combined with any type of optimization e.g. w/ morphing based shape optimization



Free Shape Optimization

- DSHAPE card

Format

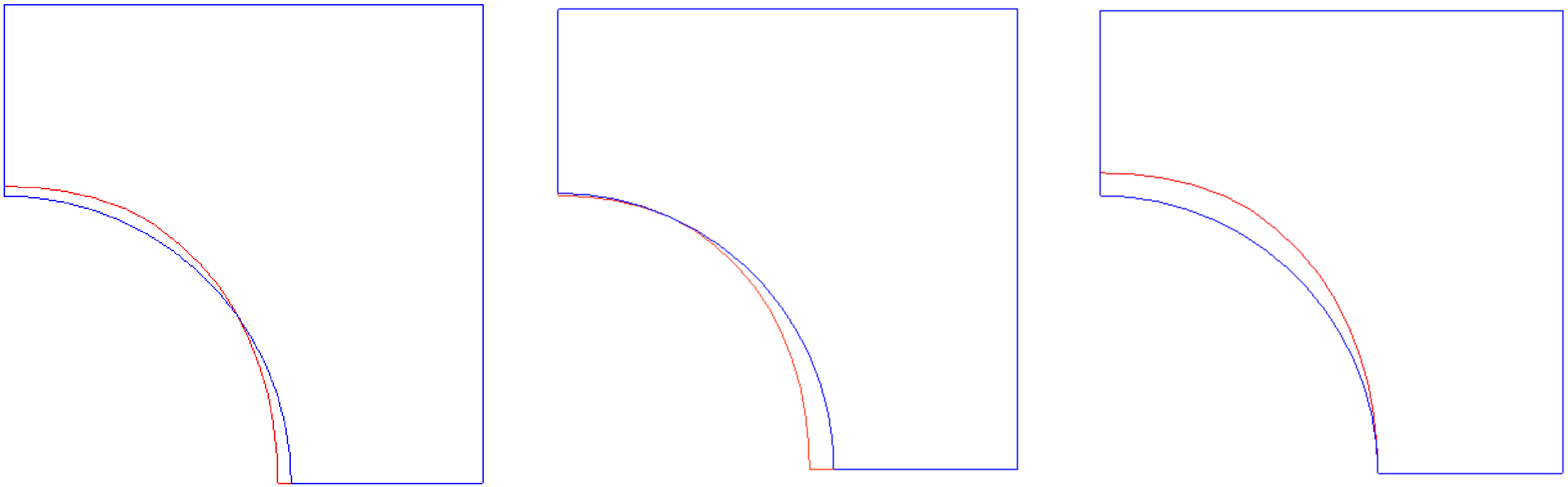
(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
DSHAPE	ID								
	PERT	DTYPE	MVFACTOR	NSMOOTH					
	GRID	GID1	GID2	GID3	GID4	GID5	GID6	GID7	
		GID8	GID9	….	….				

Optional continuation line for grid constraints

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)
	GRIDCON	GDID1	CTYPE1	CID1	X1	Y1	Z1	
		GDID2	CTYPE2	CID2	X2	Y2	Z2	
		….	…..					

Free Shape Optimization

- DTYPE



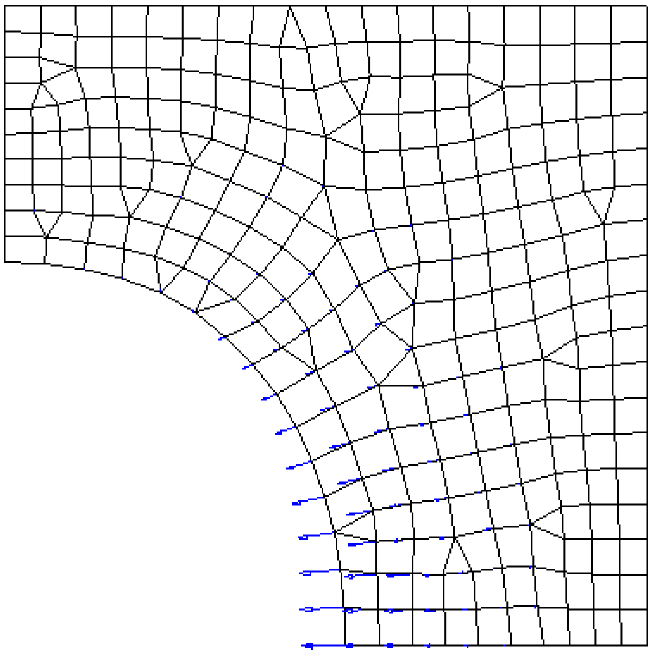
BOTH (default) GROW SHRINK

Undeformed

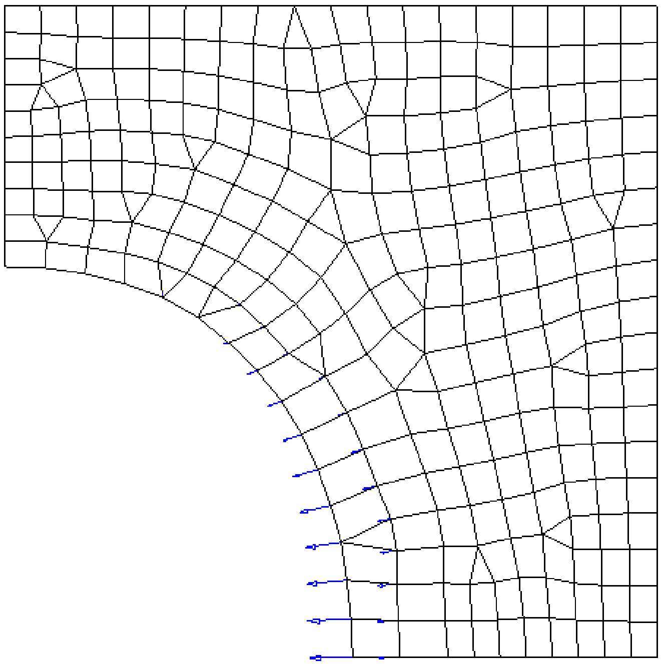
Deformed

NOTE : It's better to use 'BOTH', unless you are required to constrain the design boundary to grow or shrink

Free Shape Optimization



(a) NSMOOTH = 5

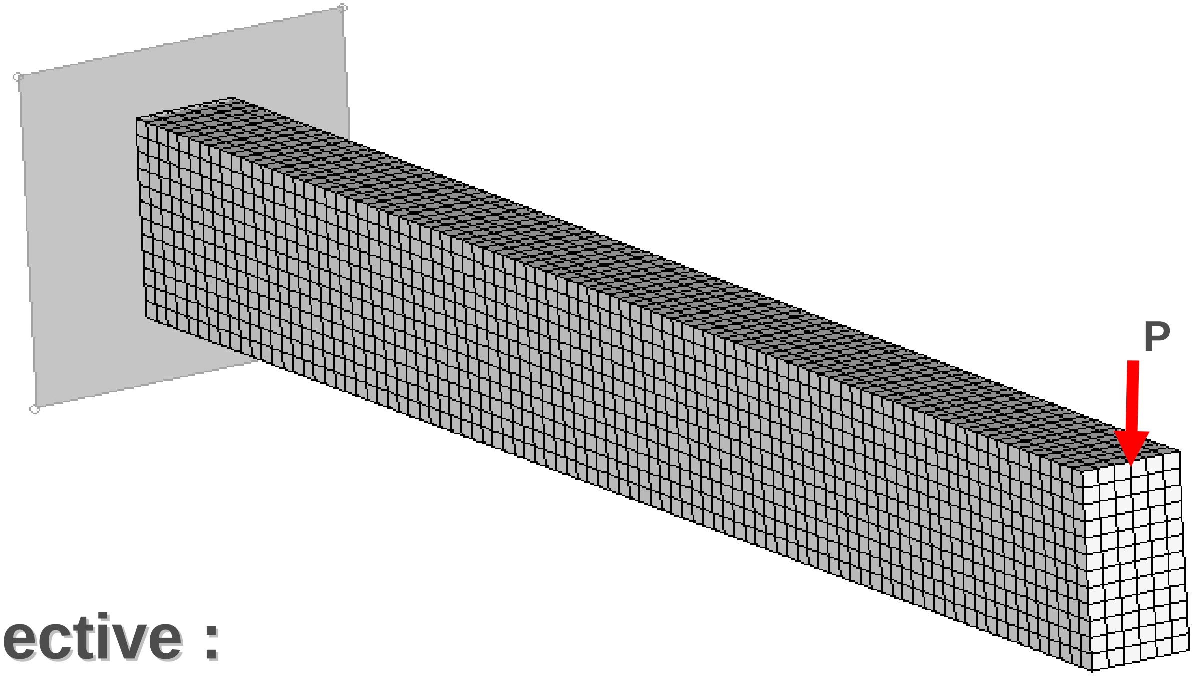


(b) NSMOOTH = 1

- Larger NSMOOTH → better in avoiding element distortion BUT slower;
- NSMOOTH can be larger than the number of available layers.

e.g., NSMOOTH = 100 will work fine in the above example.

Free Shape optimization



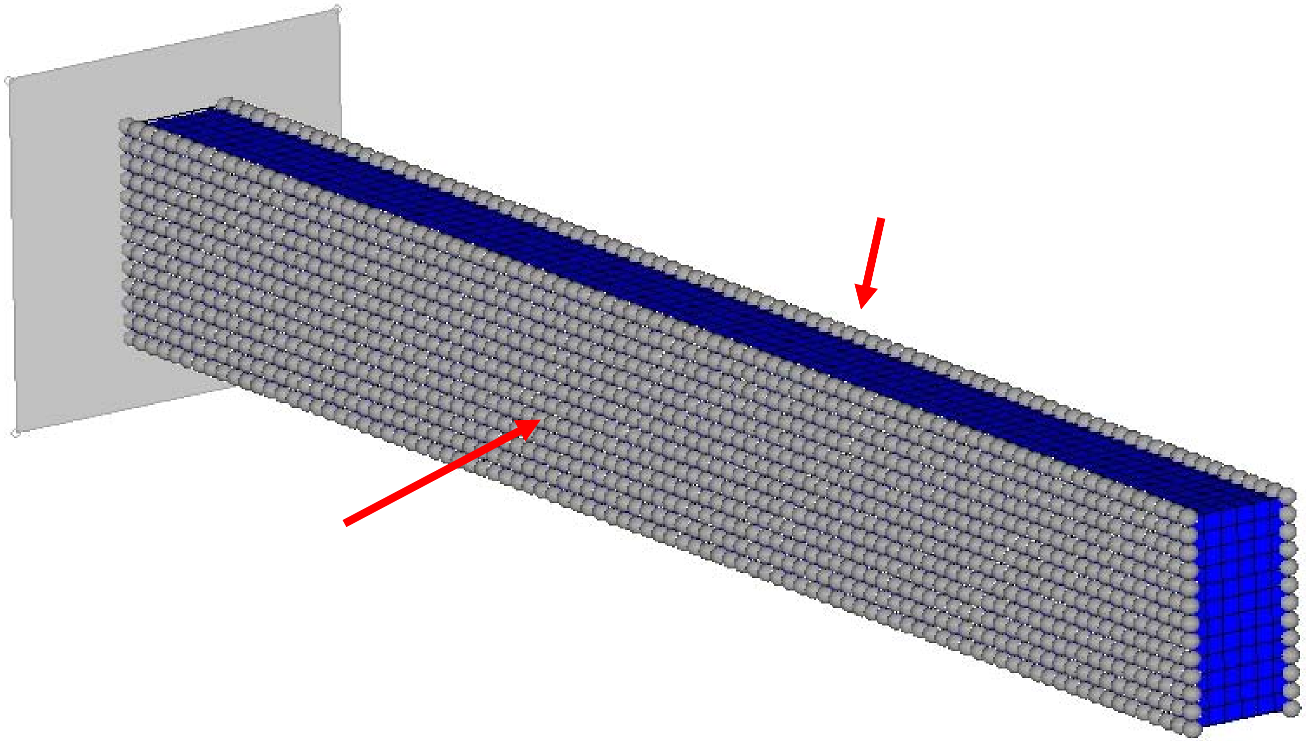
Objective :

Minimize compliance

Subject to:

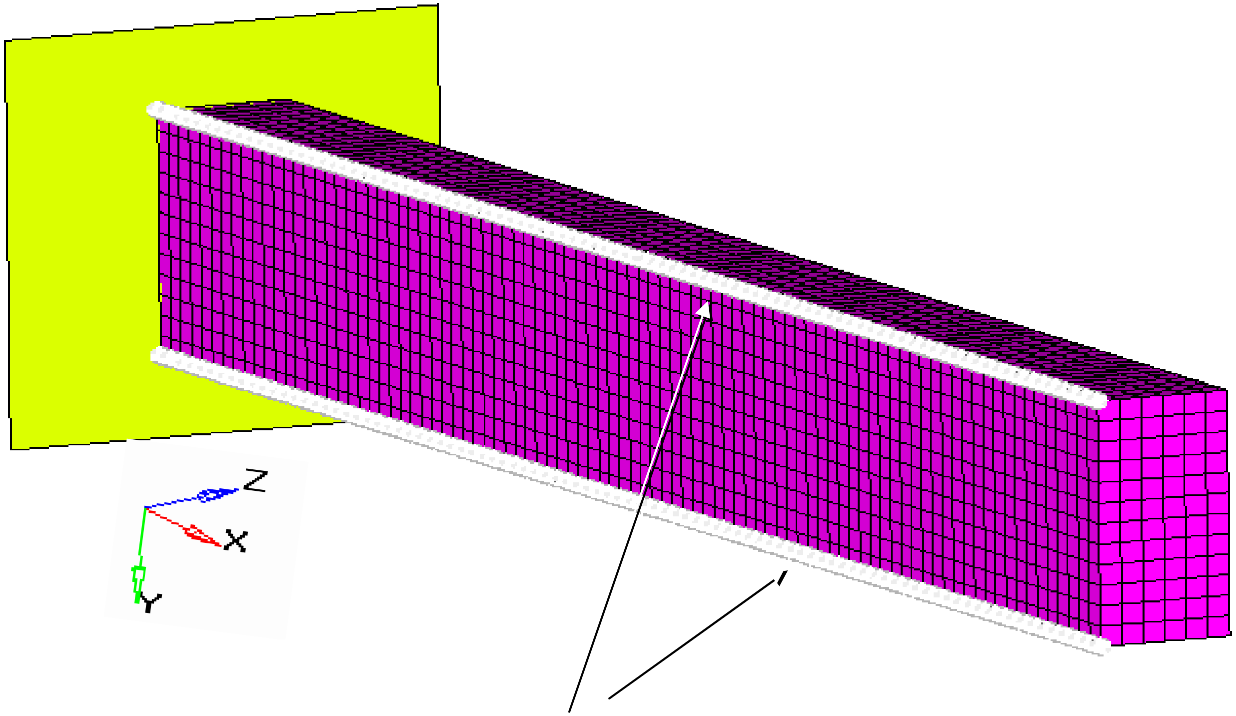
Volume < 4000.00

Free Shape optimization



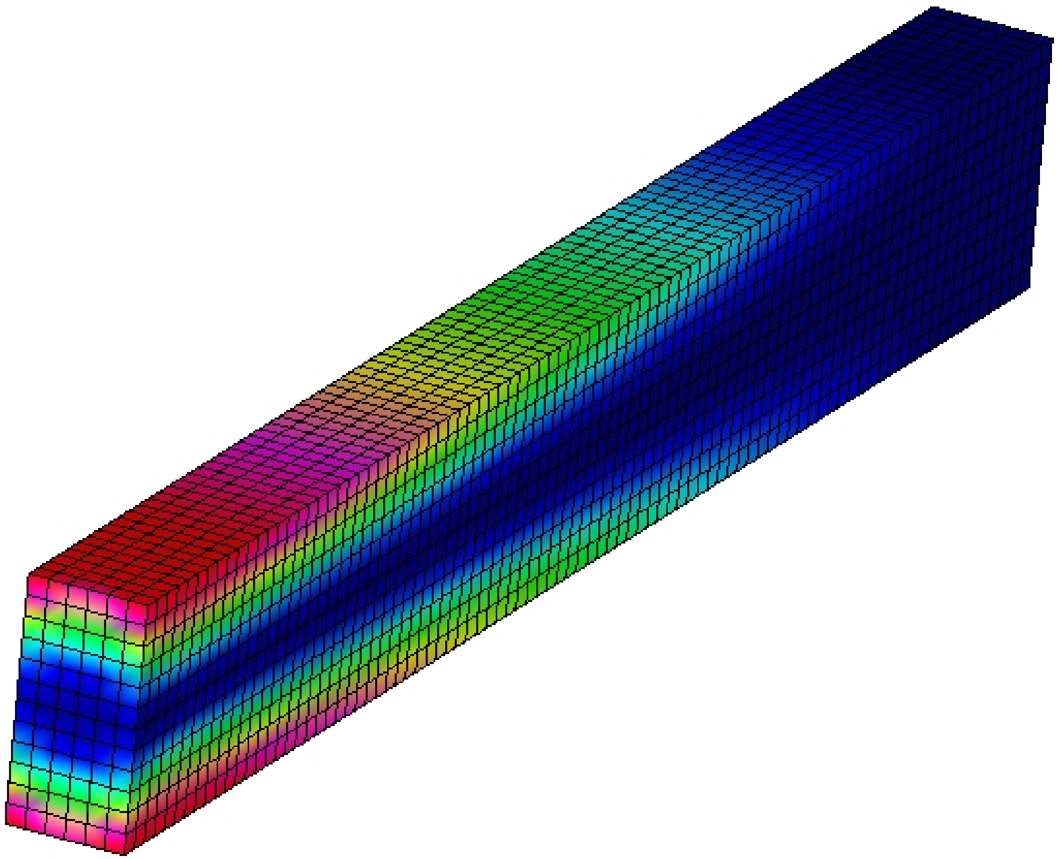
Select Free Shape design grids

Free Shape optimization



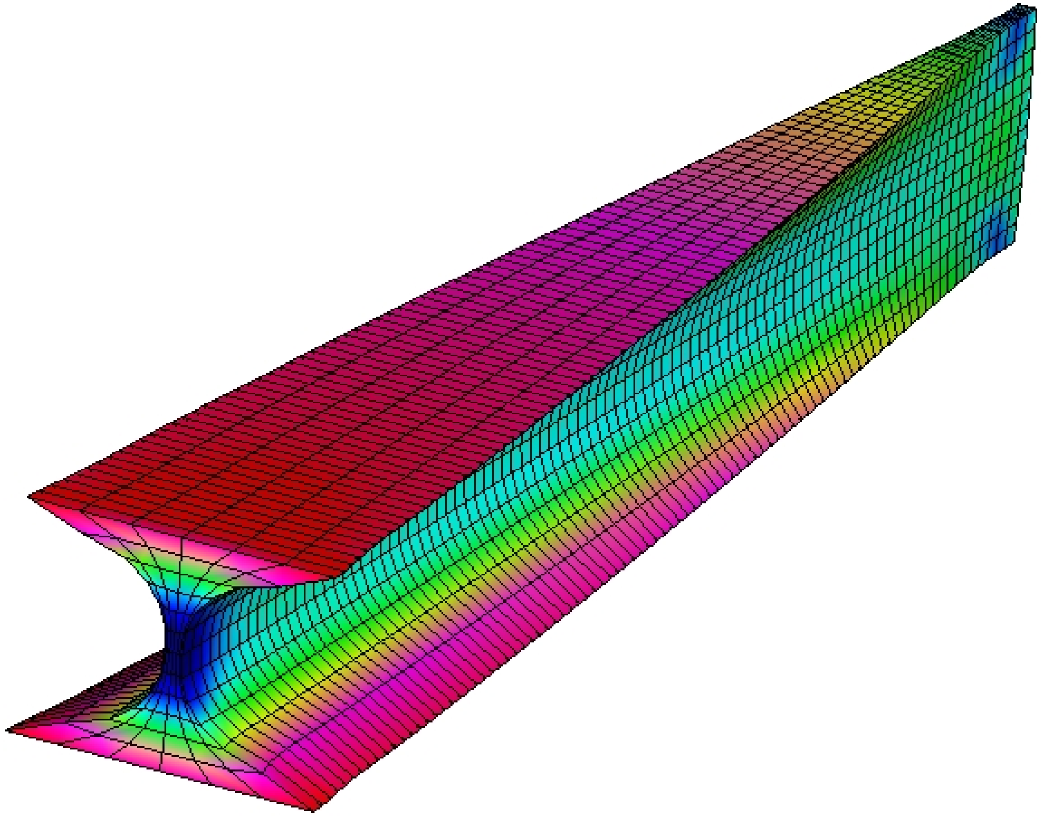
Move only on X-Z plane – fix the height of the beam section

Free Shape optimization



ITER 0 : Compliance = 4.103E+00 Volume = 6.480E+03

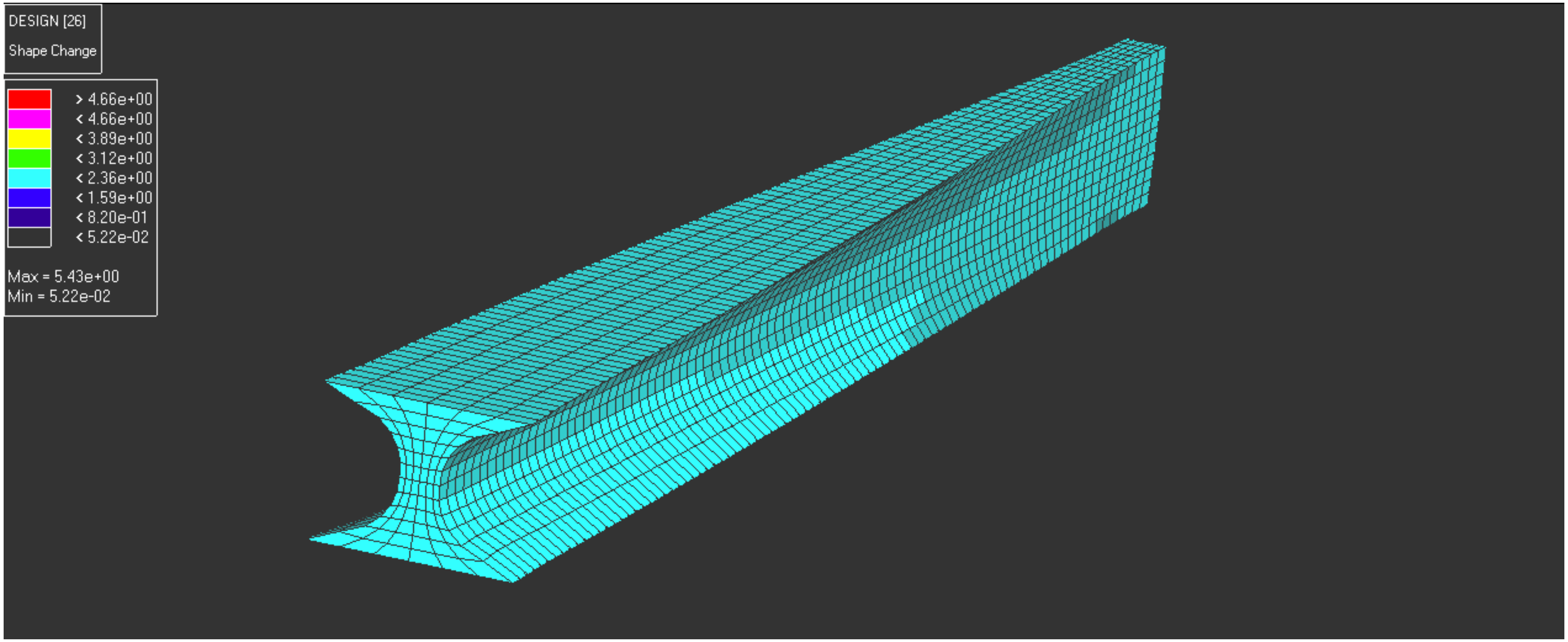
Free Shape optimization



ITER 26 : Compliance = 3.368E+00 Volume=3.994E+03

Objective -17.91%, Max. constraint violation 62.00% →0.00%

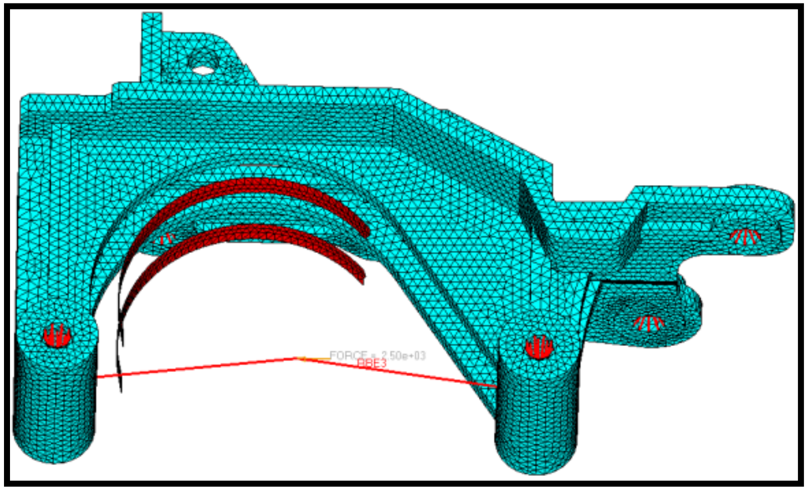
Example 2: shape change history



Shape history of the solid beam example

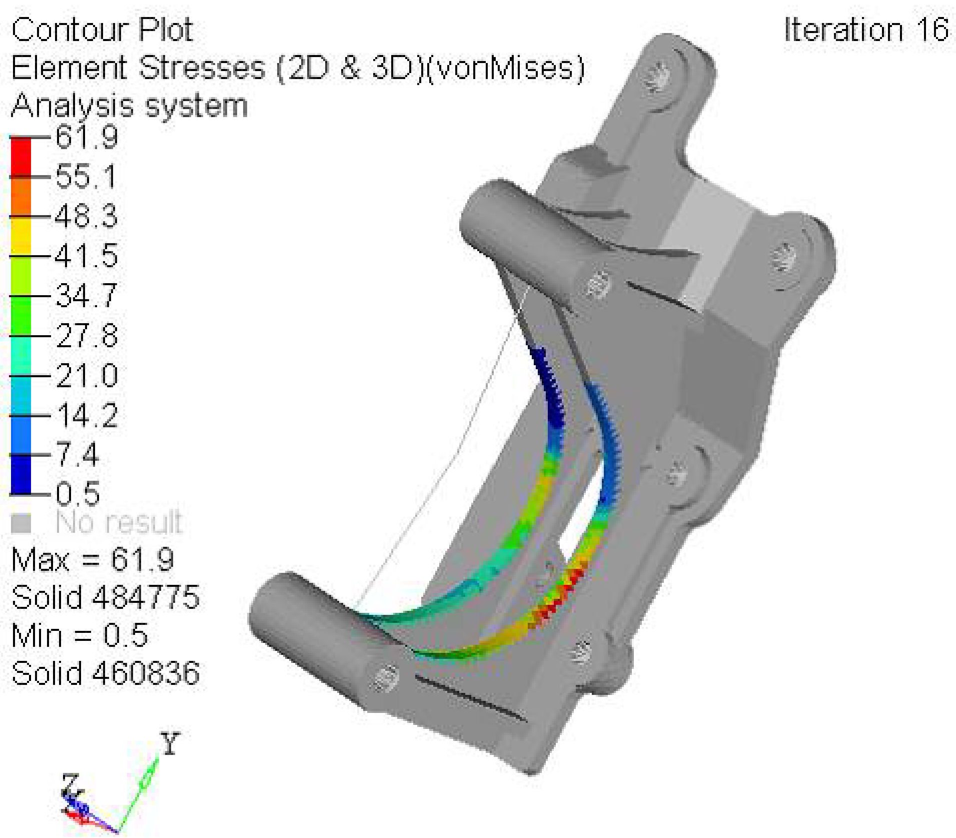
Exercise 5.3 - Free-shape Optimization Compressor Bracket

In this exercise, shape optimization on a solid model will be performed using the free-shape optimization method along with manufacturing constraints, such as symmetry and mesh barrier constraints. The objective of this optimization is to reduce the stress by changing the geometry of the model.



Objective:	Minimize mass
Constraint:	Maximum von Mises stress of the joint < 62 MPa
Design variables:	Shape variables normal to the node set selected

Exercise 5.3 - Free-shape optimization Compressor Bracket



1. Is your design objective of minimizing the mass obtained? If not, can you explain why?

2. Are your design constraints satisfied?

Maximum von Mises stress of the joint < 62 MPa

Appendix A: Composite Exercise

PHASE I - Free Size Optimization, (Ply topology)

PHASE II - Size Optimization (Thickness and number of plies)

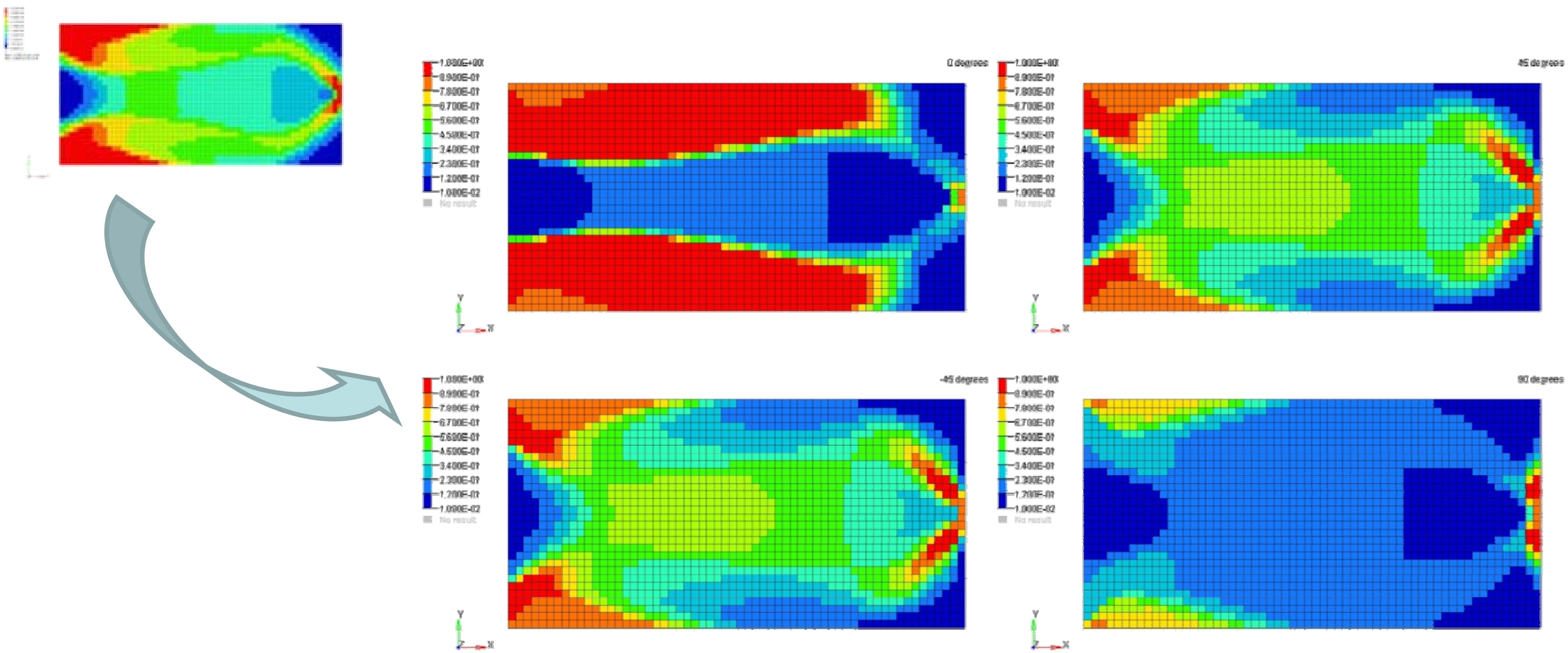
PHASE III – Shuffle Optimization (Stacking Sequence).

Composite Optimization

PHASE I - Concept

Concept: Free-Size or Topology Optimization

- Determine composite patch size, shape & location
- Incorporate manufacturing constraints



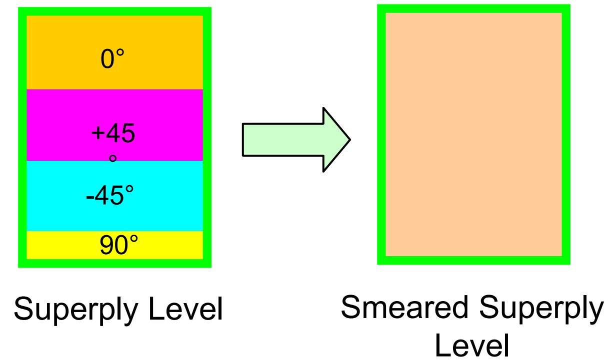
Composite Optimization

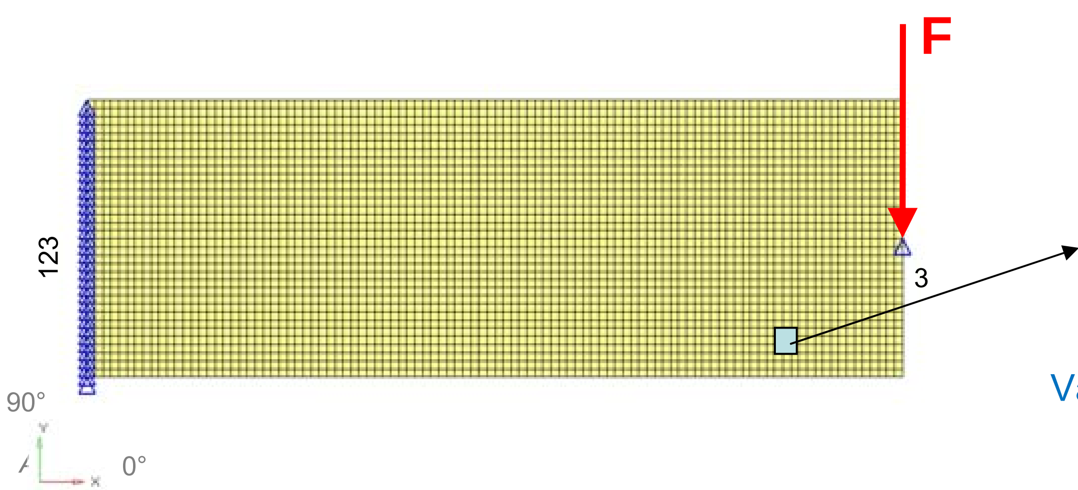
Free Size Optimization

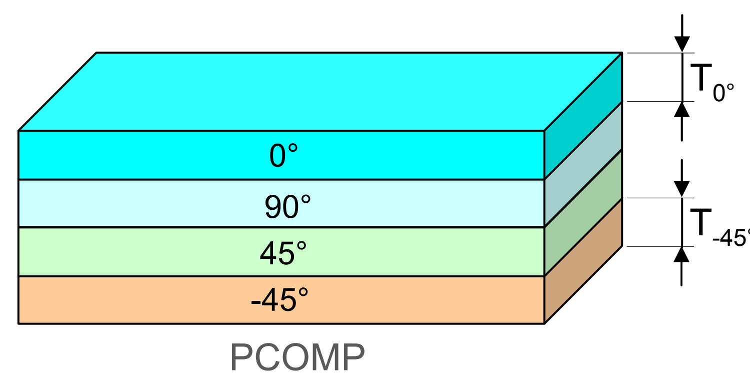
Optimization Setup

- Min (Mass)
- Maximum Displacement (u) on Tip u ≤ 0.6
- Manufacturing Constraints
  - Balanced ±45° Plies
- Design Variables Ply Thickness Ti for each Element
  - ‘Ti’ varies continuously between 0 and Ti-initial
  - If no stiffness is needed for 90°Ply in Element X, the variable T90° will reduce or become zero.
  - Additional plies with different angels can also be used.

SMEAR-PARAMETER SET



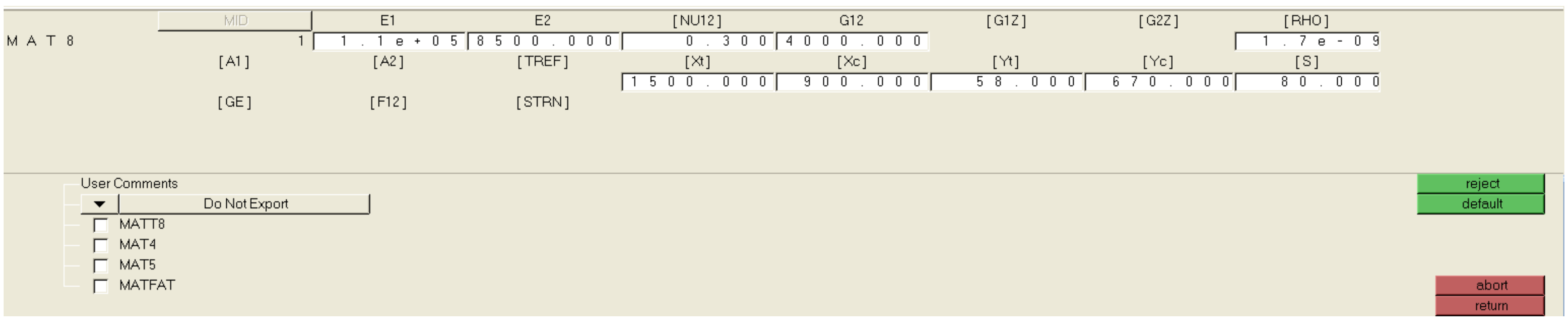




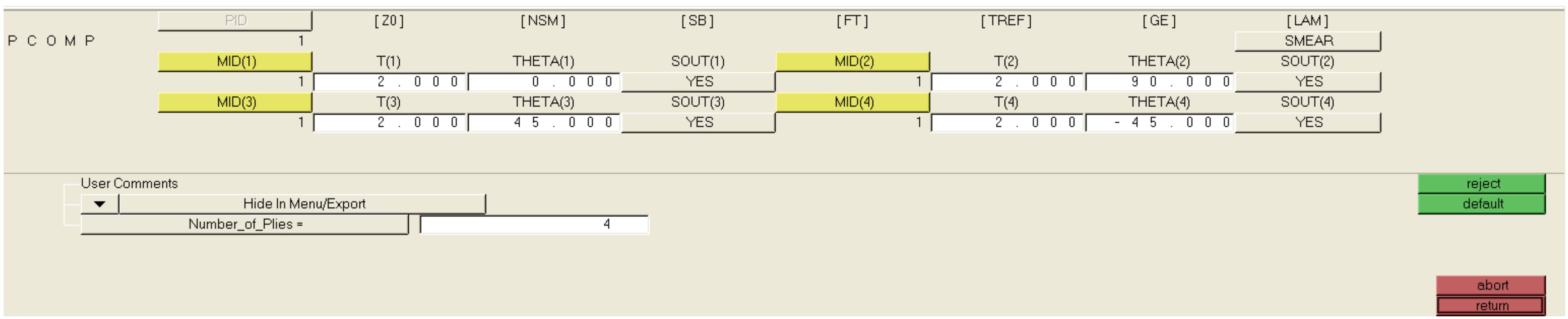
Variable: ‘T_i’ of each ‘*Super-Ply-Element*’

Composite Optimization

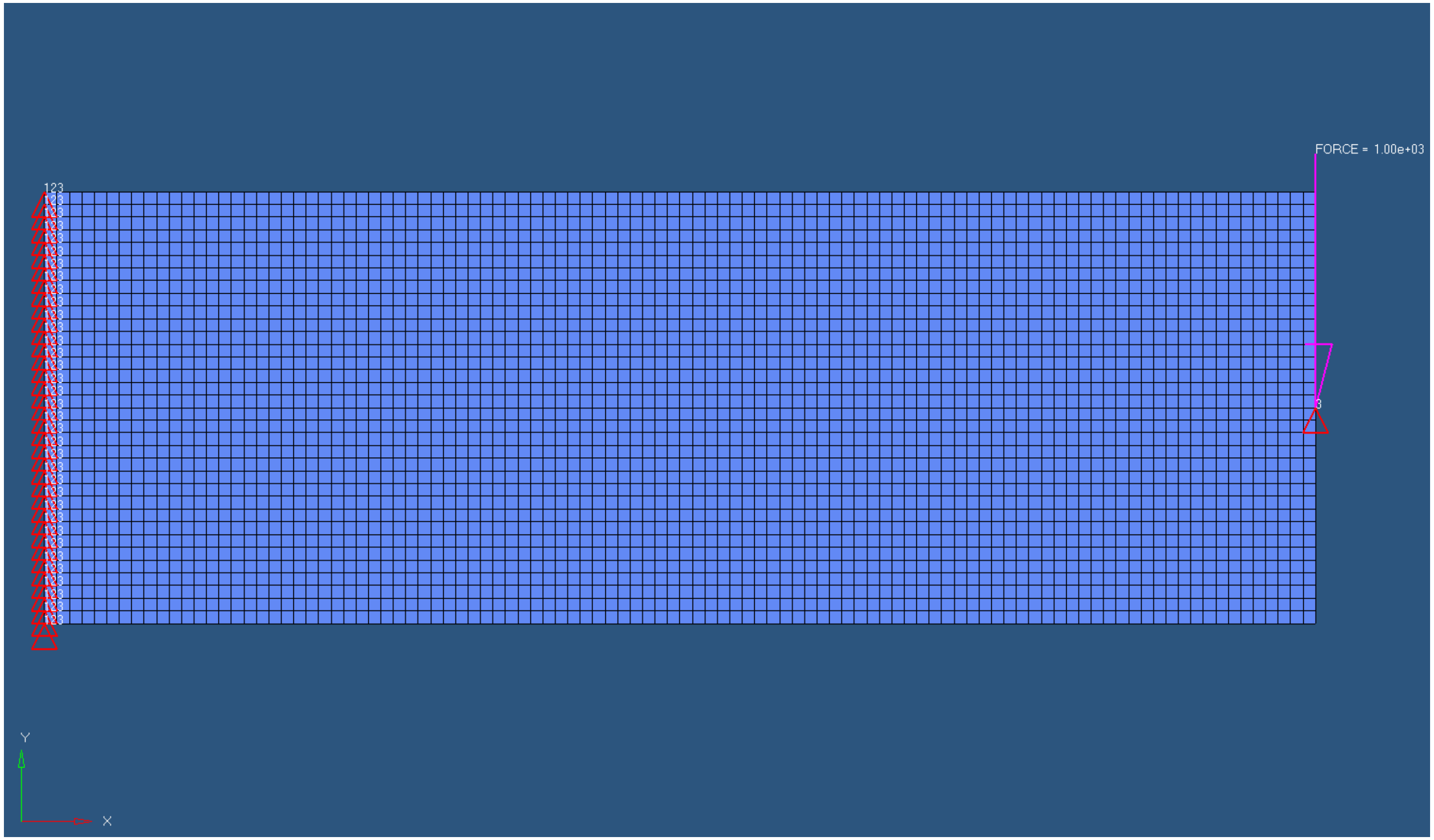
Material Definition



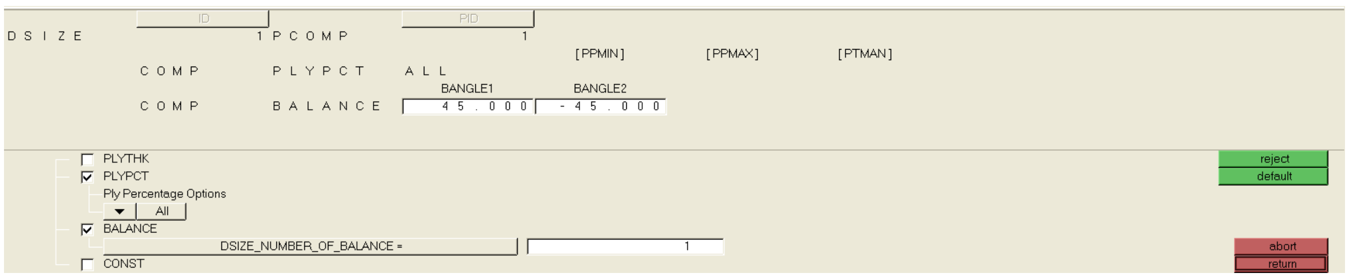
Property Definition



Finite Element Model



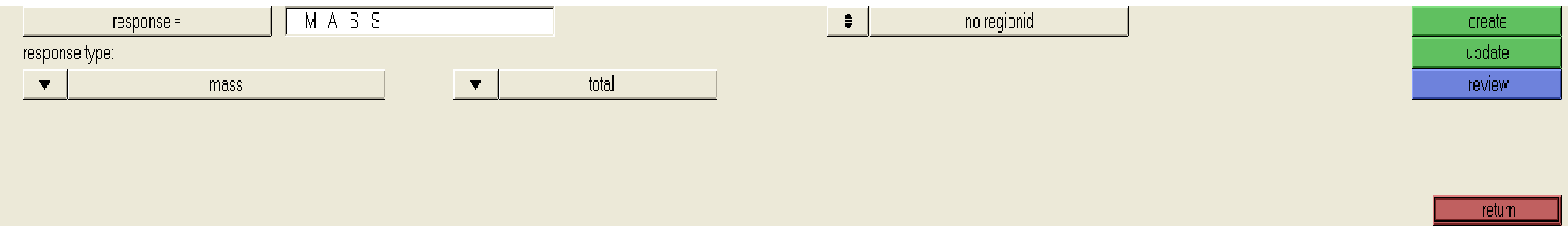
Design Variable Definition
With Manufacturing Constraints



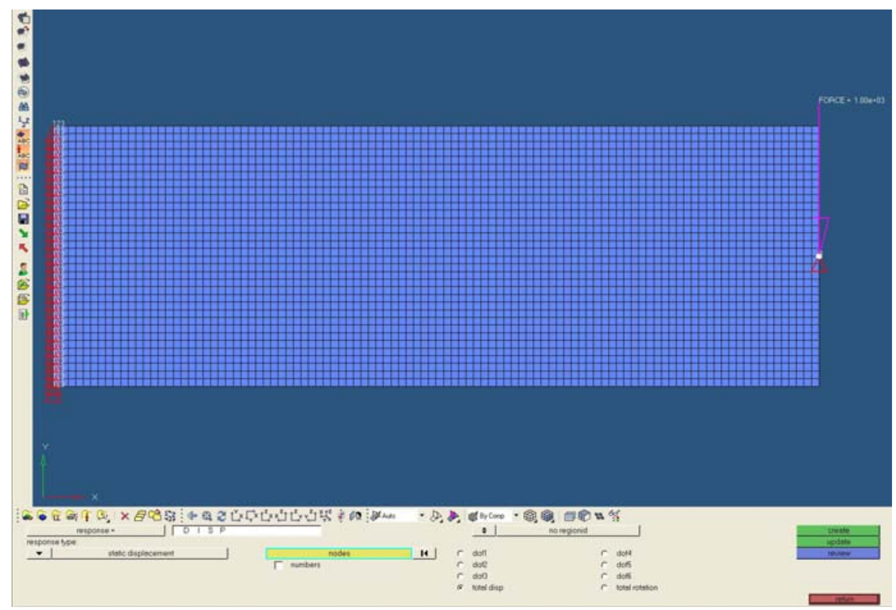
- DSIZE
    - Free Size Design Variable Definition

Optimization Responses Definition

- Total Mass

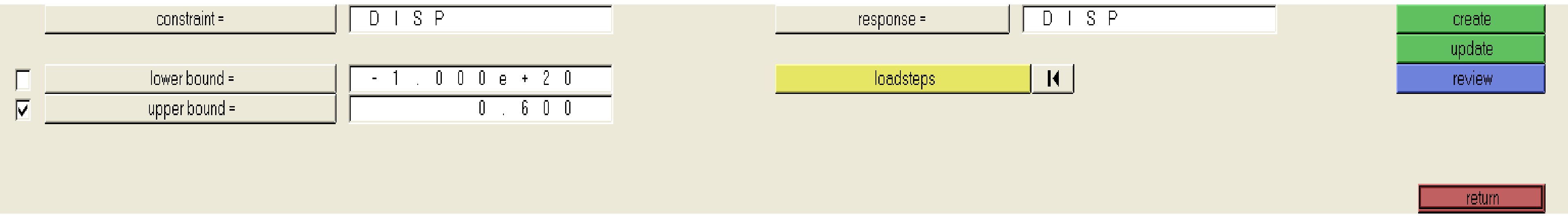


- Static Displacement
  - Total Disp
  - Node ID 2669

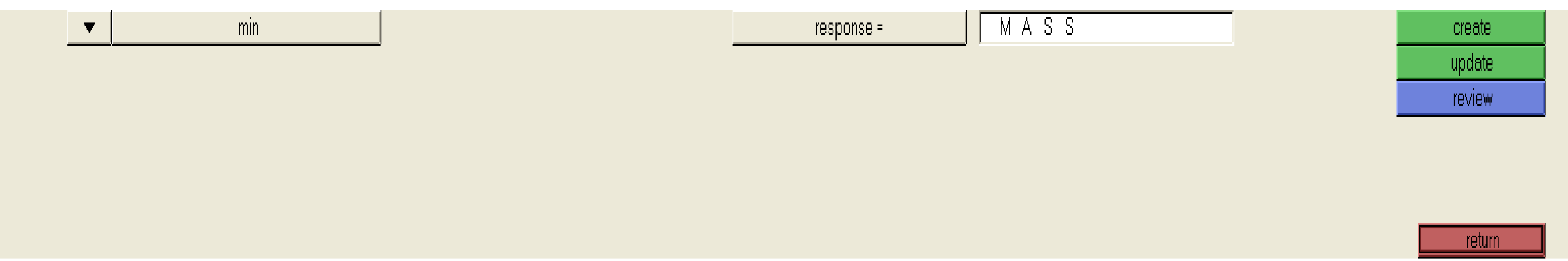


Optimization Setup

- Design Constraints



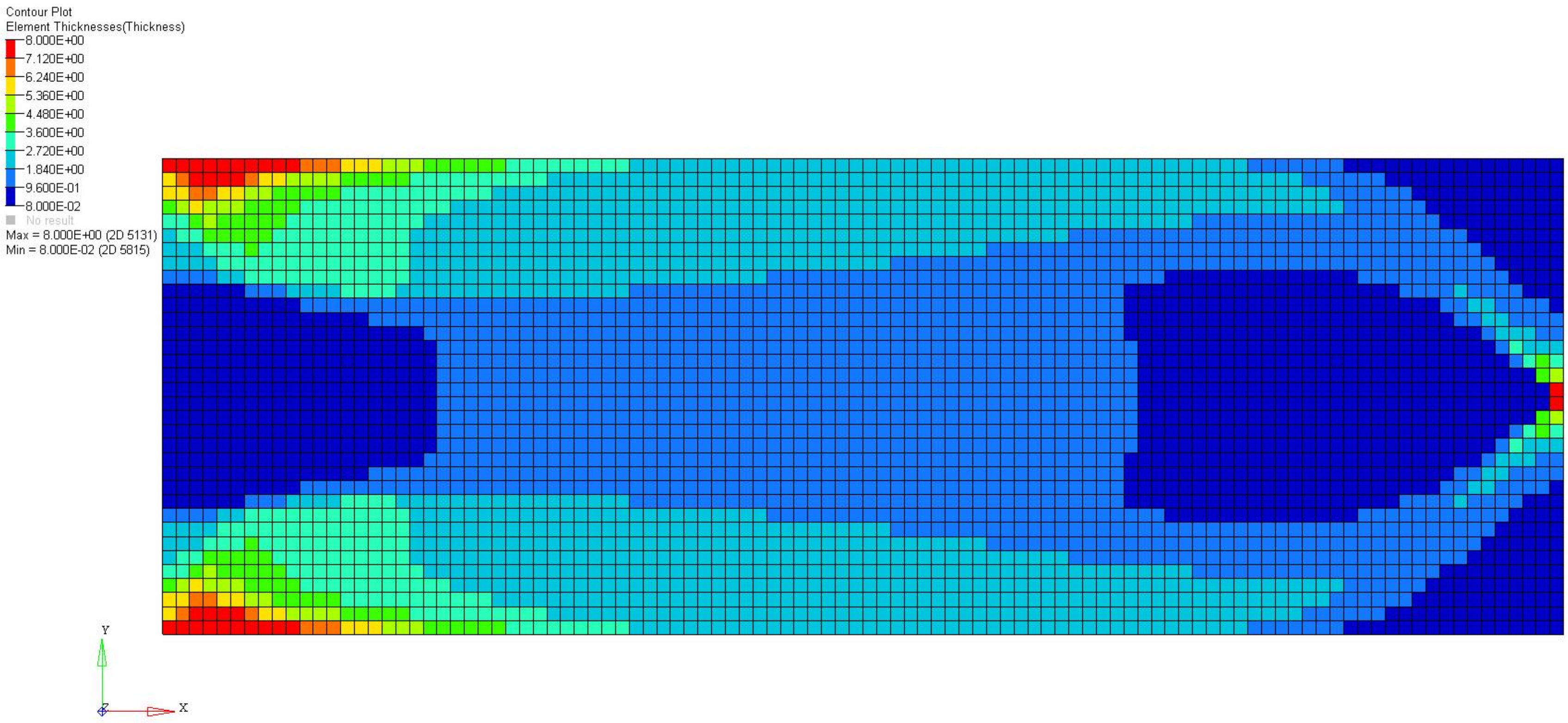
- Objective Function



Composite Optimization

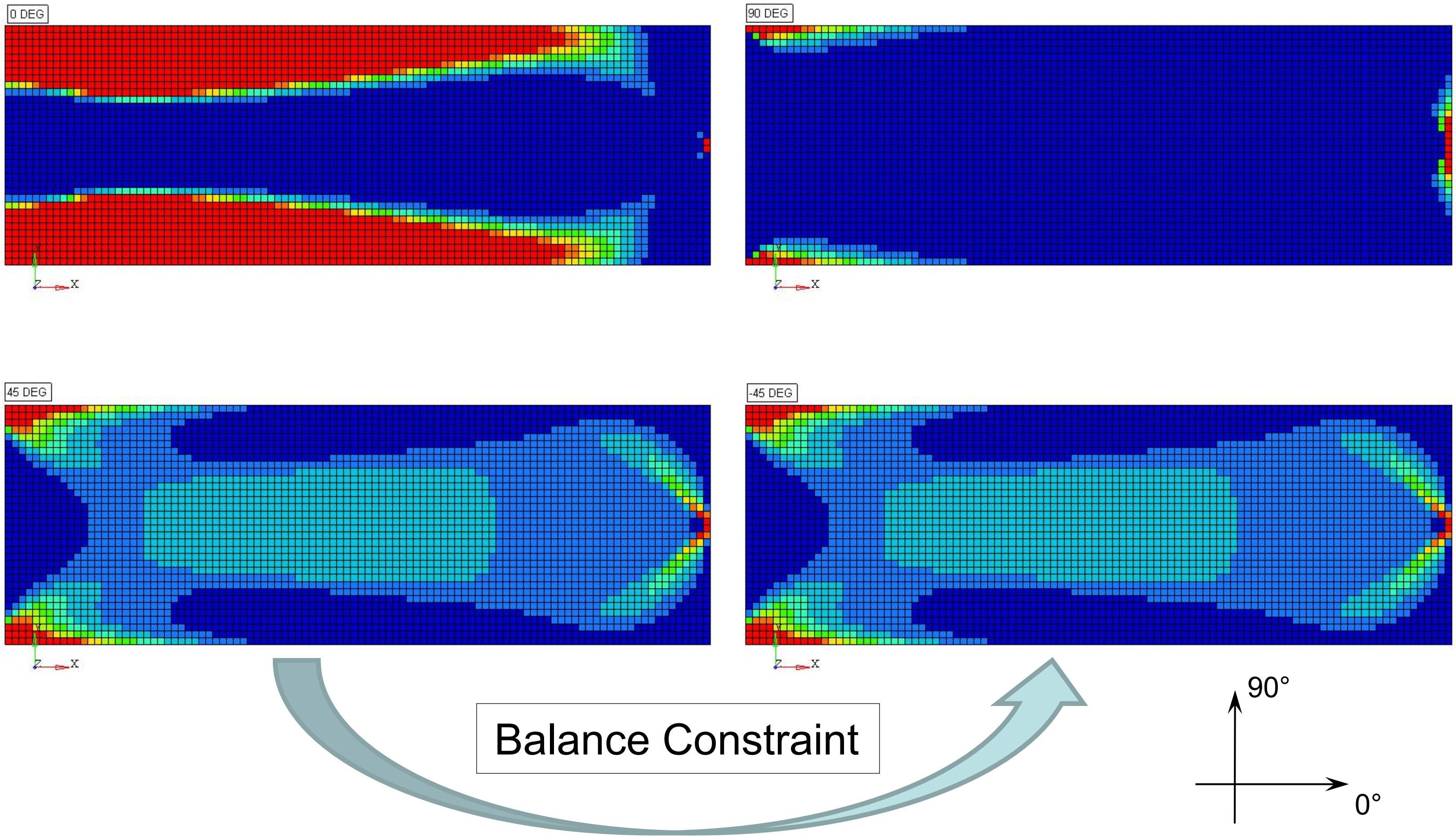
Free Size Optimization Results

Total Element Thickness Distribution



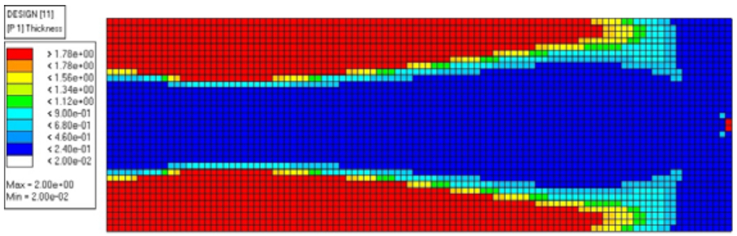
Free Size Optimization Results

Ply Thickness Distribution

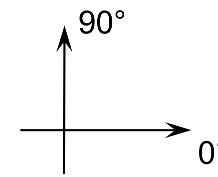


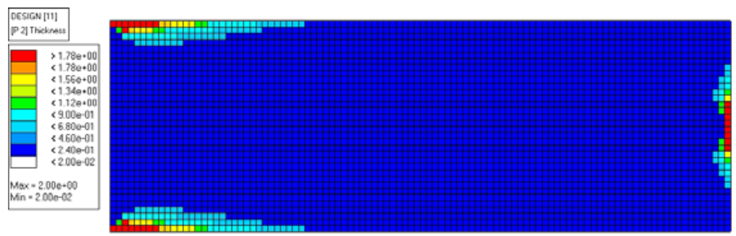
Composite Optimization

Free Size Optimization Design Proposal

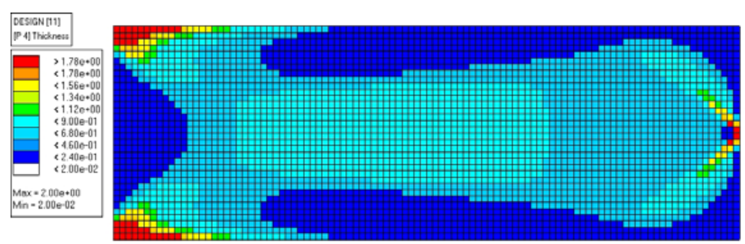


Element Thickness
Ply1 (0°)





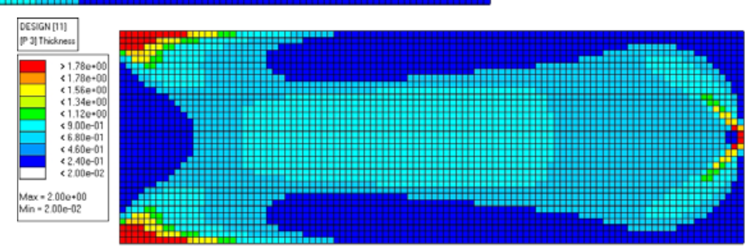
Element Thickness
Ply2 (90°)



Element Thickness
Ply3 (+45°)

LINKED by Constraint

Element Thickness
Ply4 (-45°)



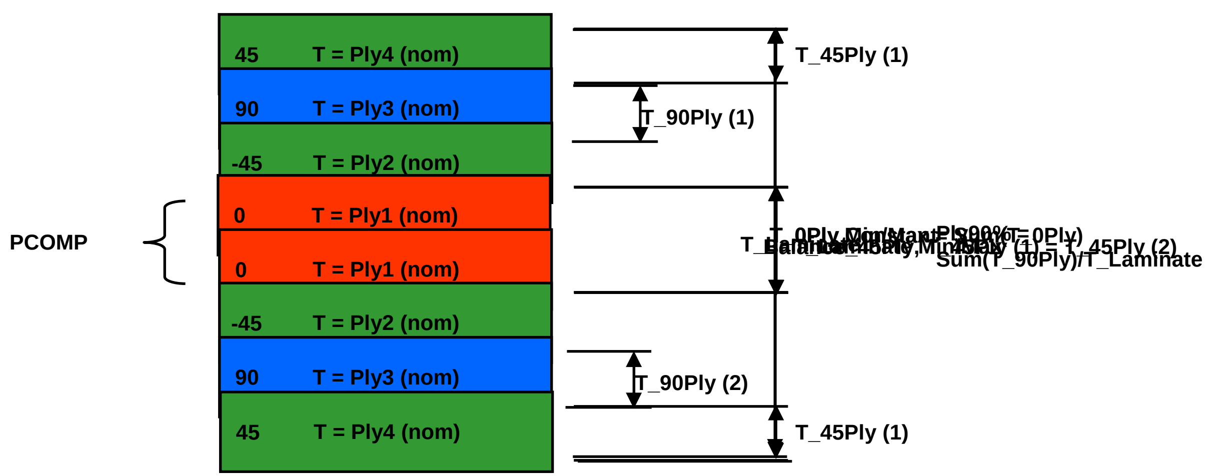
Composite Free-Size: Manufacturing Constraints

- Min. and max. total laminate thickness
- Min. and max. ply thickness
- Min. and max. percentage of a fibre orientation
- Linkage of thicknesses of plies
- Constant thickness for a particular ply orientation

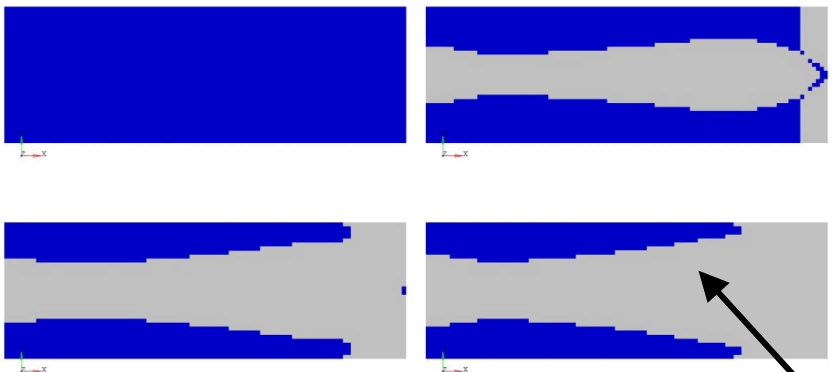
Composite Optimization

Composite Manufacturing Constraints

- Min/Max Total Laminate Thickness *(LAMTHK)*
- Min/Max Individual Ply Thickness (i.e. Min/Max 0-Deg Thickness…) *(PLYTHK)*
- Min/Max Individual Ply Angle Percentage (i.e. %90...) *(PLYPCT)*
- Balanced Ply Angles (i.e. Balance +/- 45's) *(BALANCE)*
- Constant Individual Ply Thickness *(CONST)*

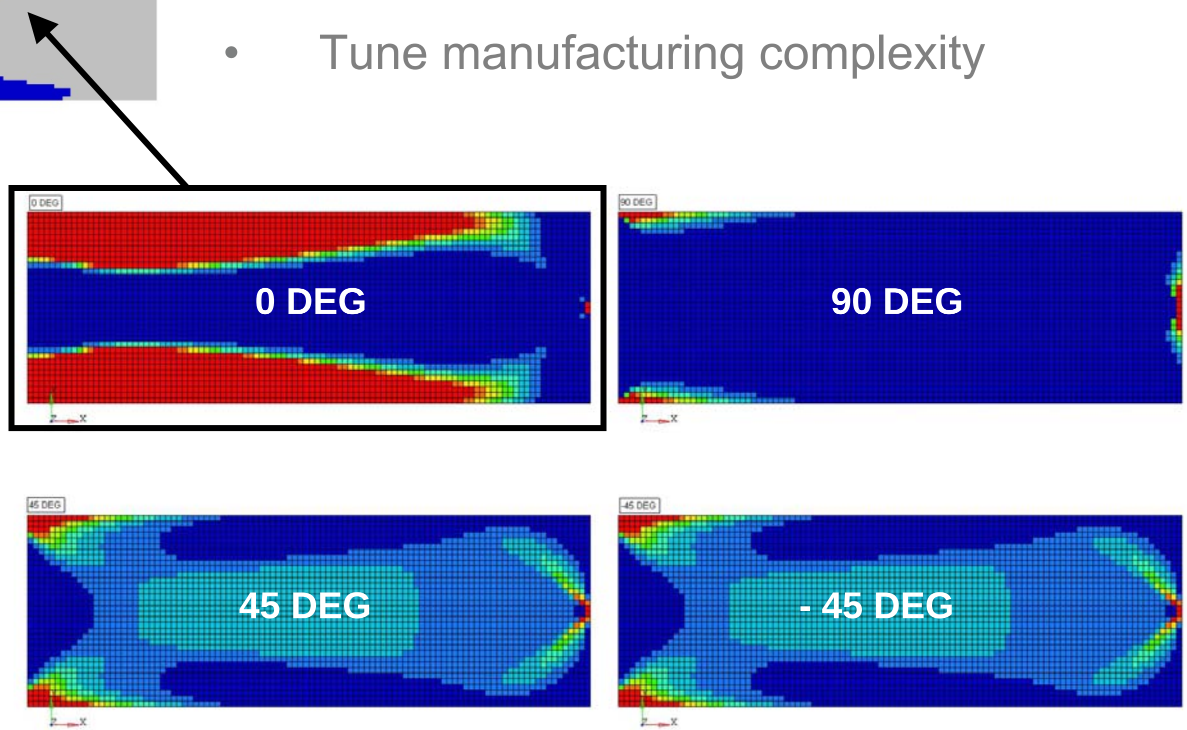


Automatic Generation of Plies for Sizing

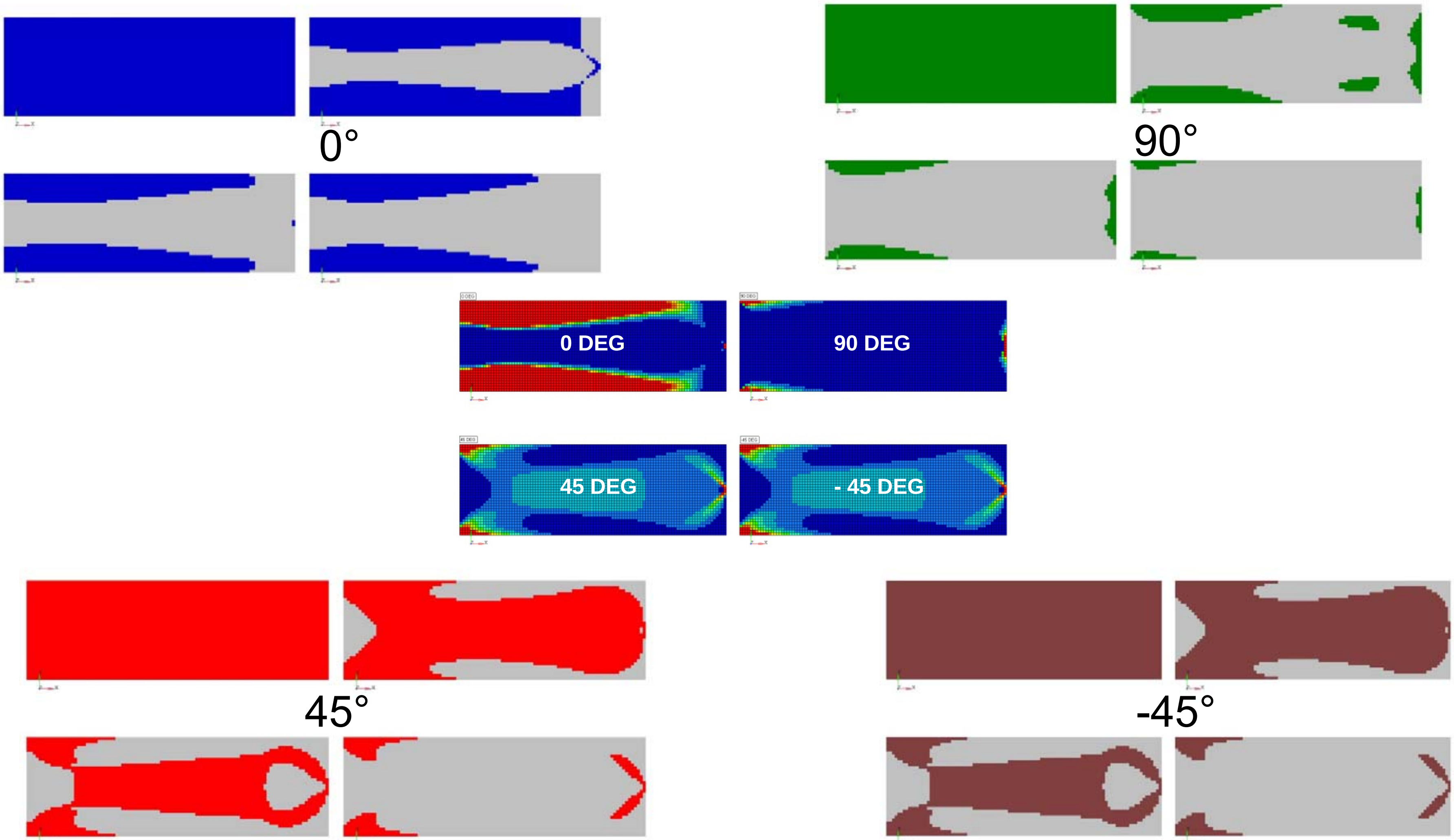


E.g. 4 Ply Bundles for 0°

- Automatic extraction of plies from free-sizing optimization
- User defined number of ply bundles per ply orientation
- Tune manufacturing complexity

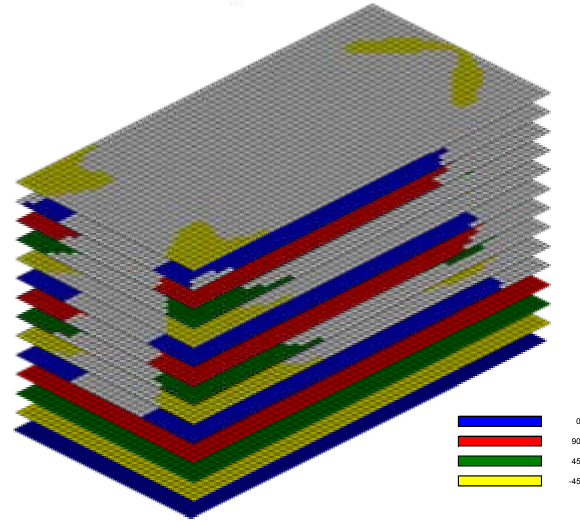


Automatic Generation of Plies for Sizing



Free Size Optimization Output Parameter

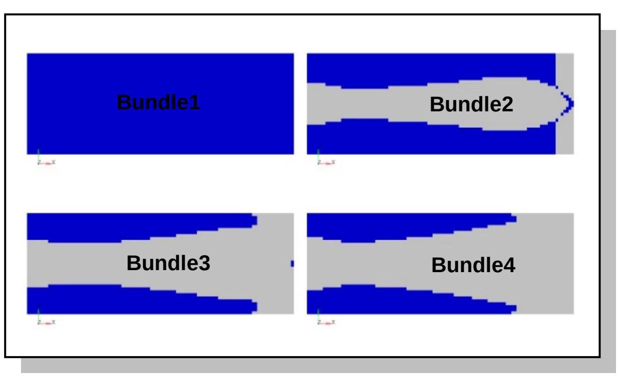
```
$$
$$ Optistruct Input Deck Generated by HyperMesh Version  : 9.0b121
$$ Generated using HyperMesh-Optistruct Template Version : 9.0b121
$$
$$   Template:   Optistruct
$$
$$
$
SCREEN OUT
OUTPUT,FSTOSZ,YES,8
$$------------------------------------------------------------------------------$
$$                         Case Control Cards                                   $
$$------------------------------------------------------------------------------$
$
$  OBJECTIVES Data
$
$
$HMNAME OBJECTIVES       2objective
$
DESOBJ(MIN)=3
$
```



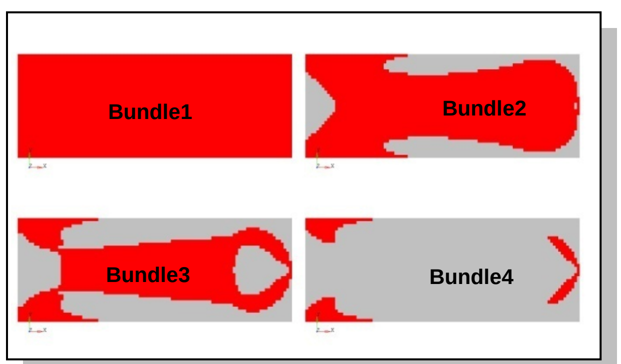
OUTPUT,FSTOSZ,YES,VALUE

- OUTPUT: Creates a new *.sizing.fem deck
- FSTOSZ: "FreeSize to Size"
- YES: Activate output
- VALUE: Number of ply bundles (Default = 4)

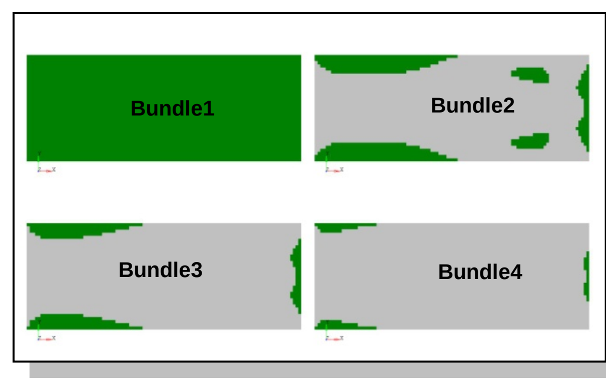
Level setting Ply-Bundles: 0° plies



Level setting Ply-Bundles: ±45° plies



Level setting Ply-Bundles: 90° plies



Free Size to Size Output

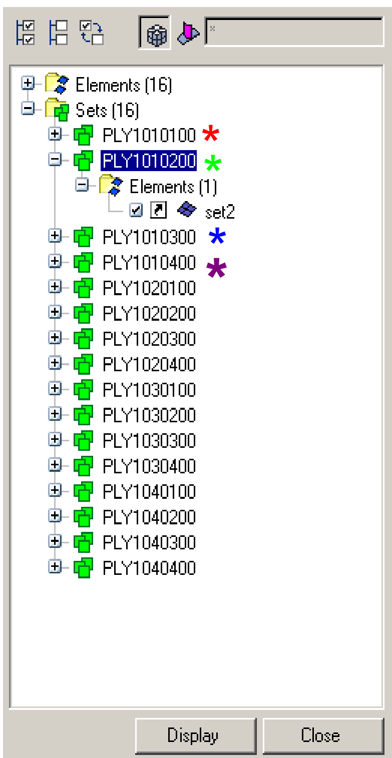
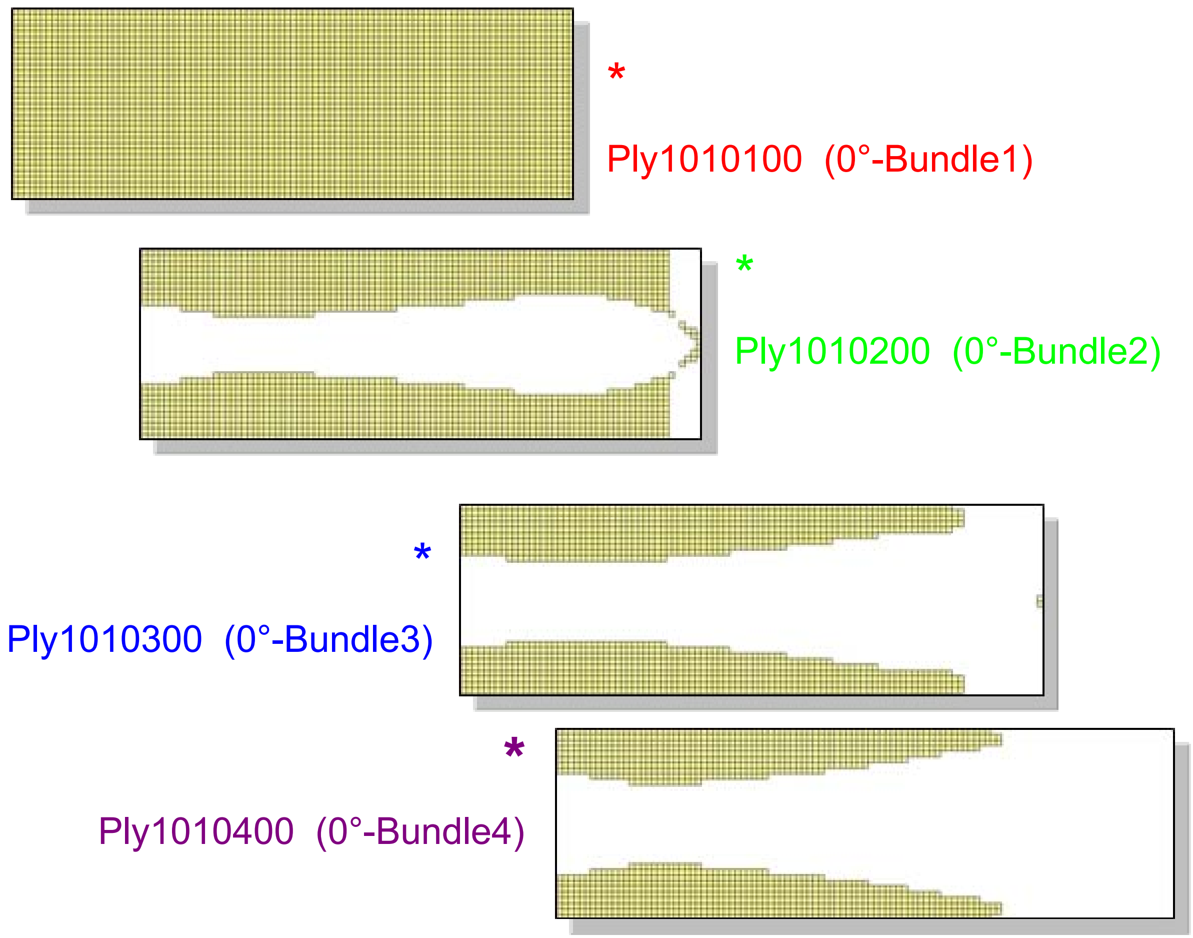
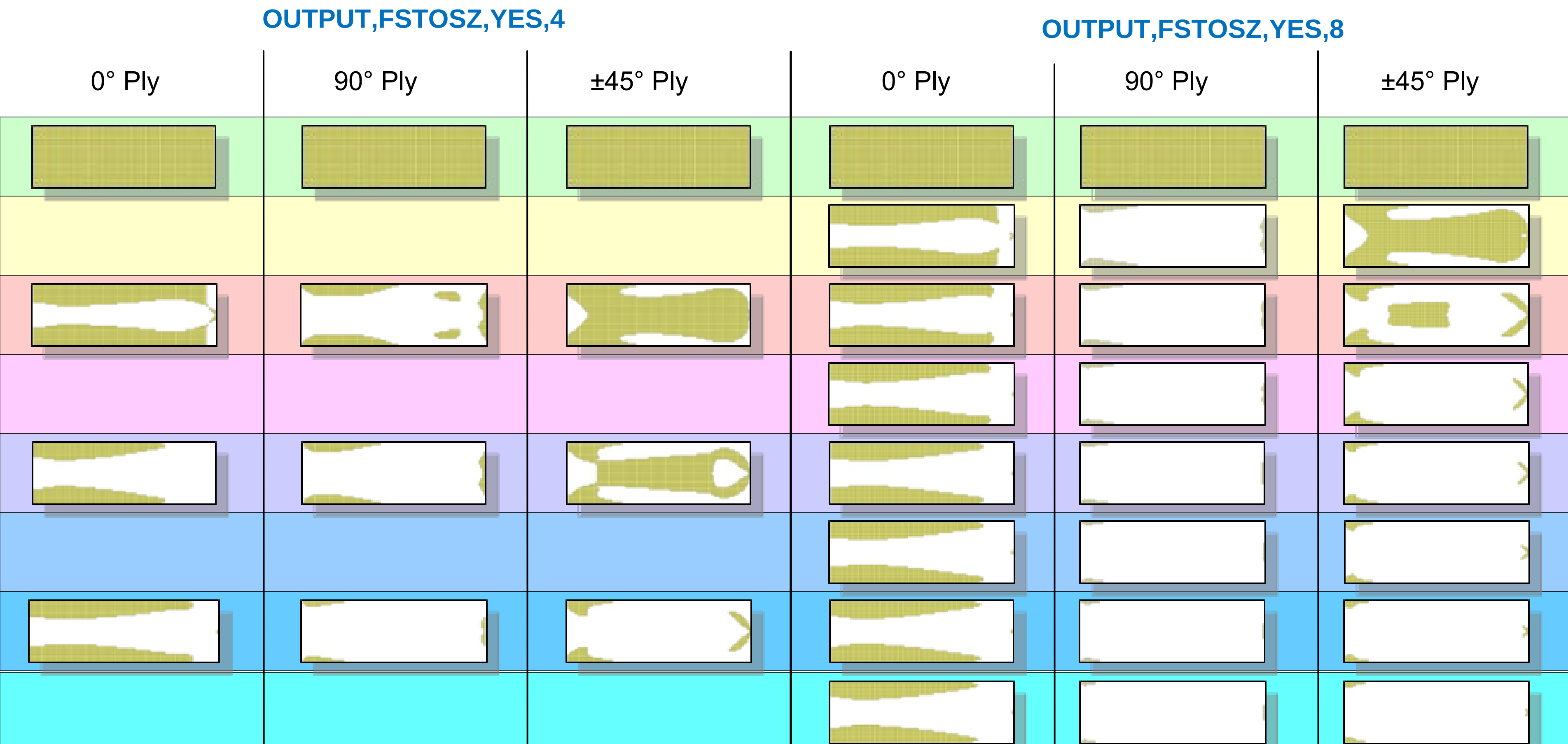


Fig. “Set Browser”

Example 0° Plies



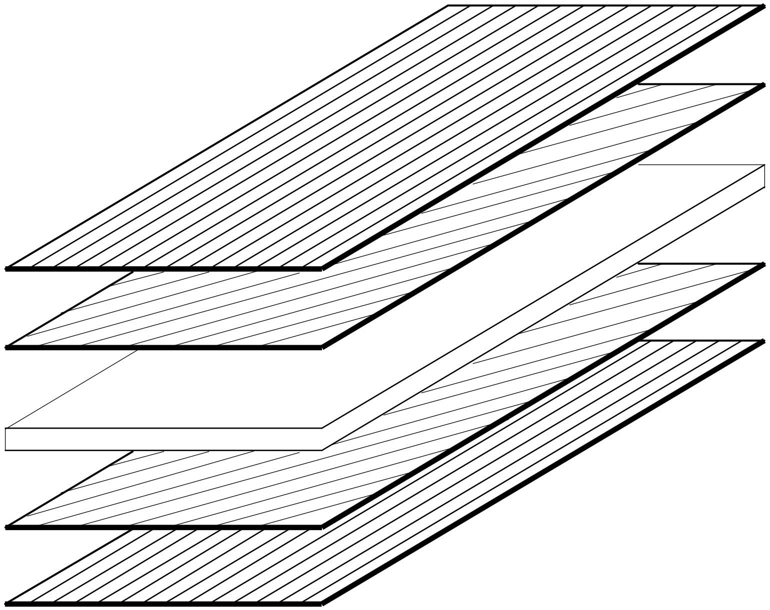
Result trade off: Cost driven vs. Weight driven



PHASE II – Dimension

System: Ply-Bundle Sizing with ply-based FEA modeling

- Determine required number of plies per patch
- All behavior constraints
- Manufacturing constraints

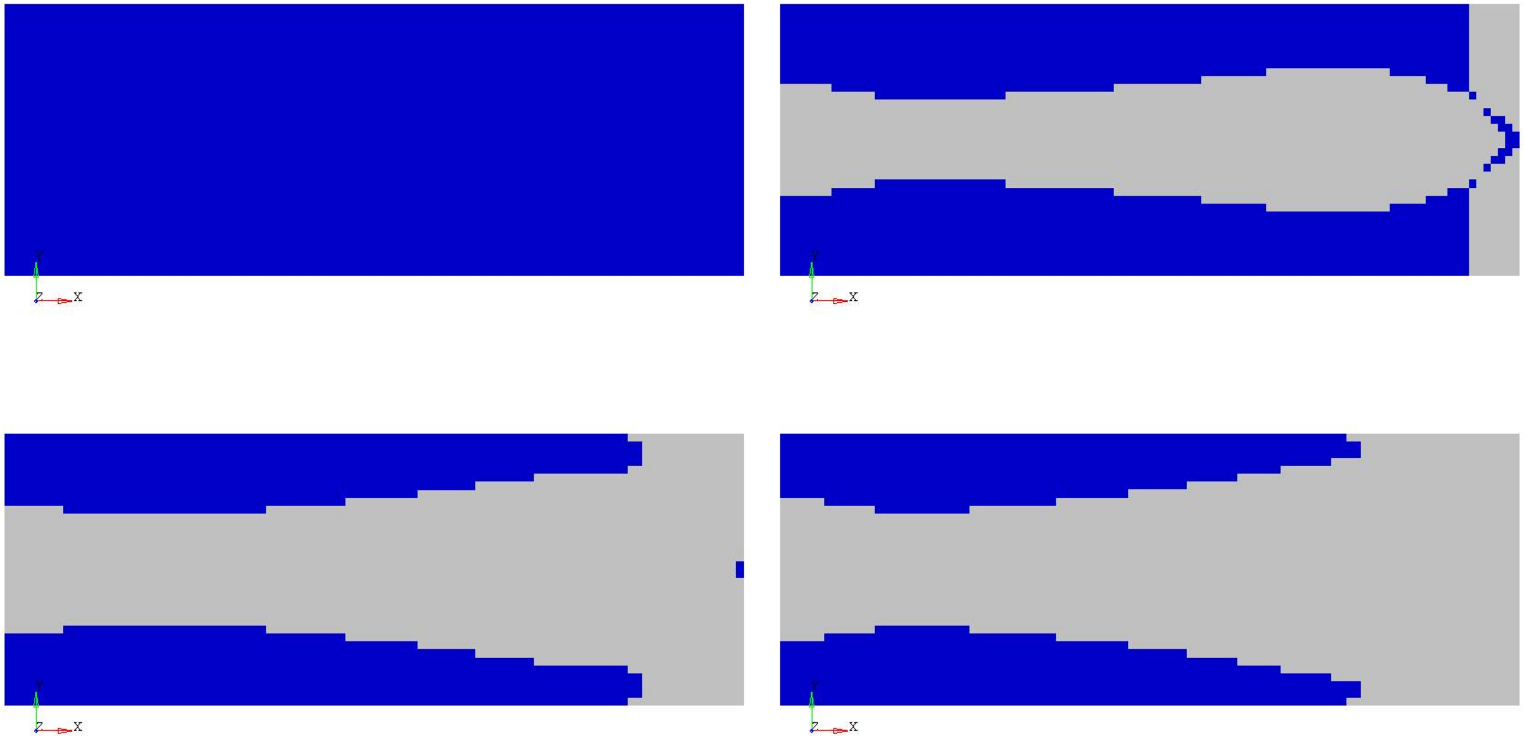


Phase II – Dimension

- Ply-Bundle Concept
  - Free-Size interpretation through level-setting thickness field
- Ply-Based FEA modeling
  - PLY
  - STACK
  - Element properties
- Ply-Bundle Sizing Optimization
  - *Discrete* optimization of Ply-Bundle thickness
  - All Behavior constraints (failure, displacement, buckling etc.)
  - Design and manufacturing constraints

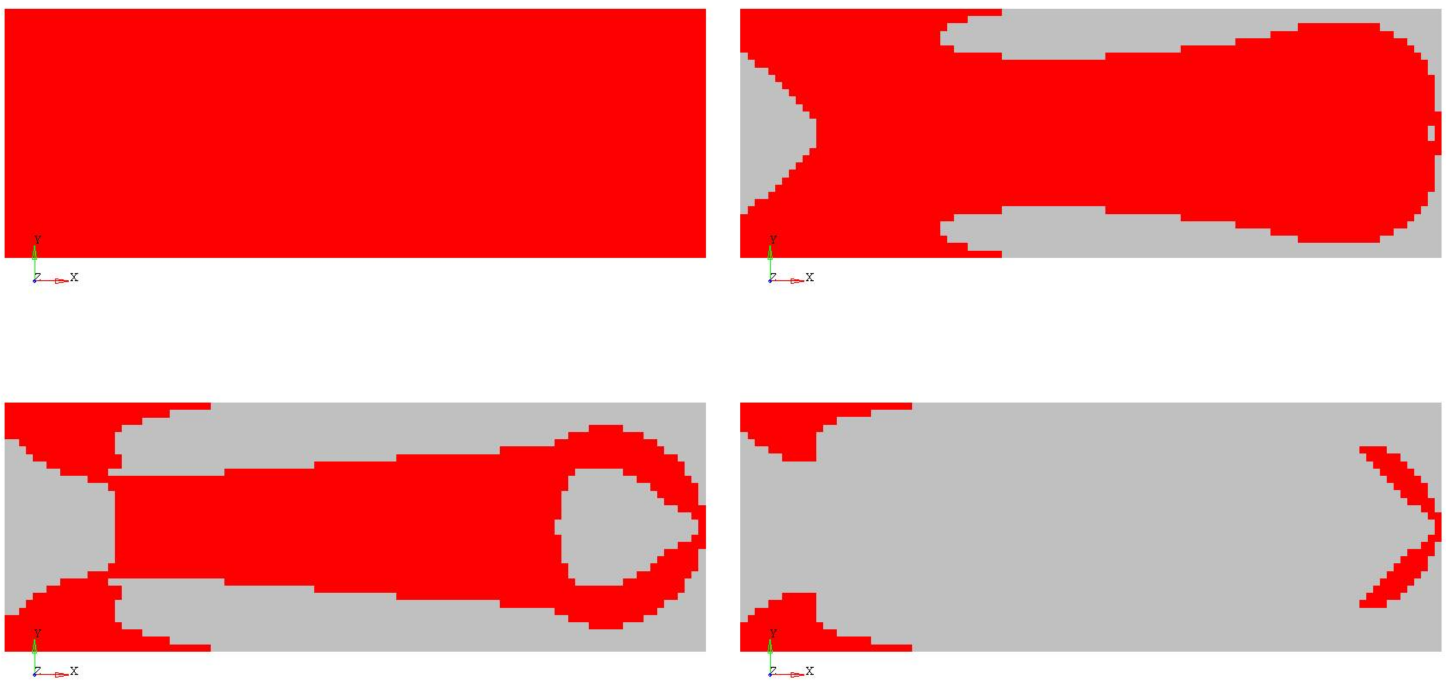
Phase II – Dimension

Level setting Ply-Bundles: 0° plies



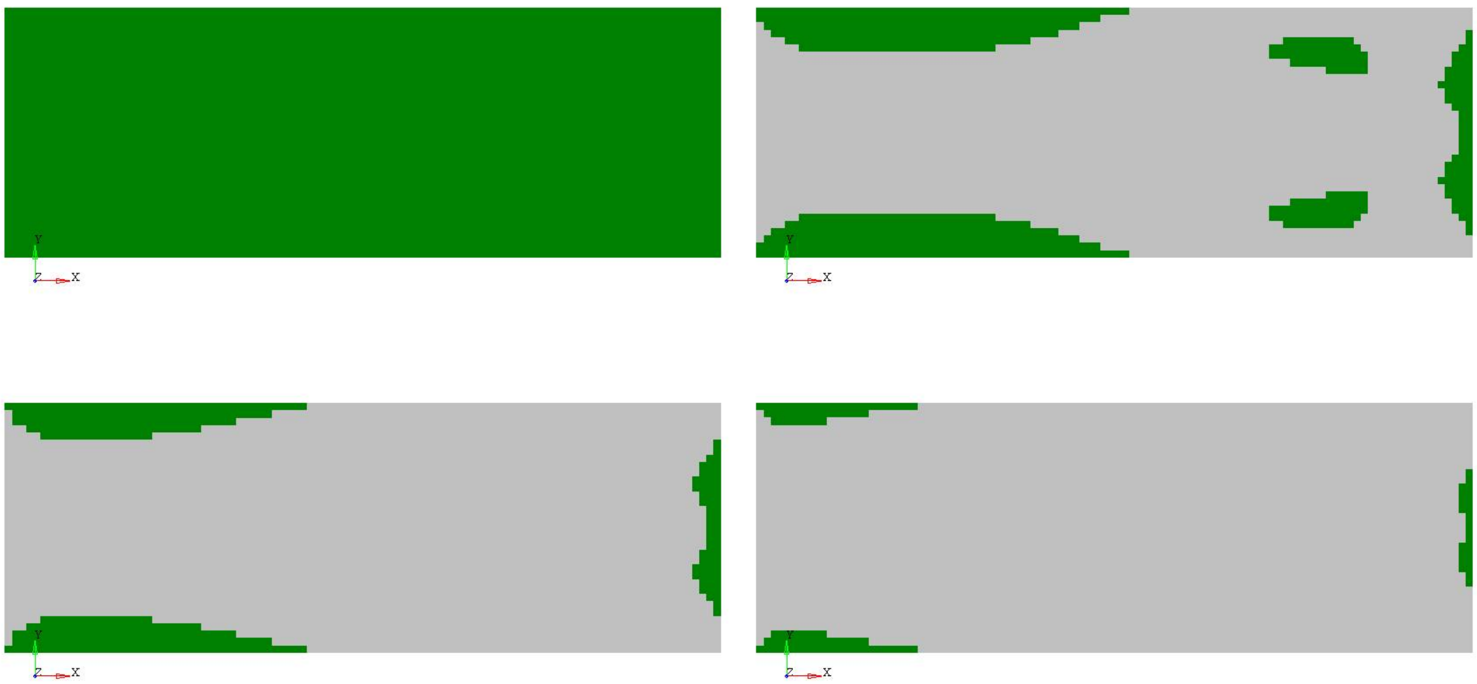
Phase II – Dimension

Level setting Ply-Bundles: +/- 45° plies



Phase II – Dimension

Level setting Ply-Bundles: 90° plies



Ply-Based FEA Modeling

- PLY – fiber orientation and layout (element sets)
- STACK ‘*glues*’ PLYs into laminate
- Element properties – *implicit* through STACK and PLYs (replacing PCOMP for explicit laminate definition)

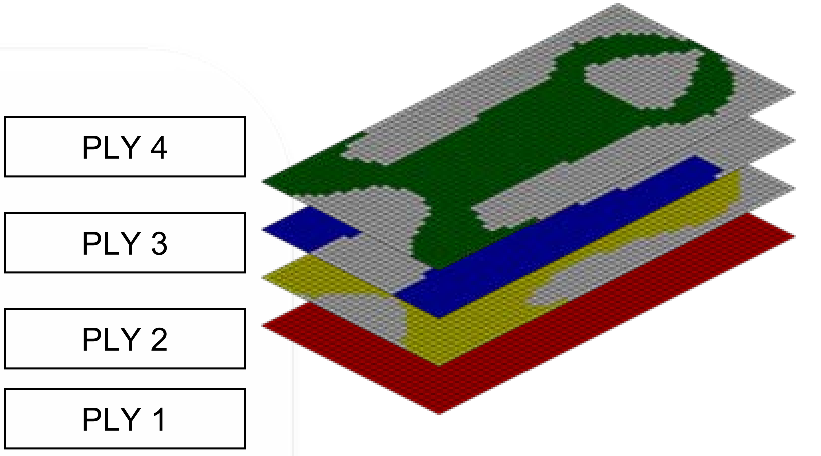
Ply	ID	MID	T	THETA	SOUT	TMANUF		
+	ESID1	ESID2	ESID3	ESID4	ESID5	ESID6	ESID7	ESID8
+	ESID9	…						

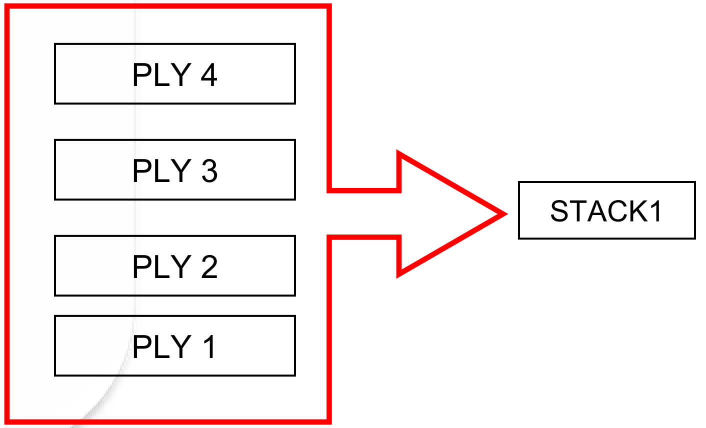
STACK	ID	LAM	PLYID1	PLYID2	PLYID3	PLYID4	PLYID5	PLYID6
+	PLYID7	…						
+								

- Native language for
  - Laminate tools (Fibersim, Anaglyph …)
  - Manufacturing – Ply-Book
  - Optimization definition

PLY and STACK Cards

```
 SET    1030200          10 "PLY1030200"
 LY     1030200 1          0.19100245.0    YES      0.125
+       10
$
$HMSET  1030300          10 "PLY1030300"
PLY     1030300 1          0.19012345.0    YES      0.125
+       11
$
$HMSET  1030400          10 "PLY1030400"
PLY     1030400 1          1.43348645.0    YES      0.125
+       12
$
$HMSET  1040100          10 "PLY1040100"
PLY     1040100 1          0.185388-45.0   YES      0.125
+       13
$
$HMSET  1040200          10 "PLY1040200"
PLY     1040200 1          0.191002-45.0   YES      0.125
+       14
$
$HMSET  1040300          10 "PLY1040300"
PLY     1040300 1          0.190123-45.0   YES      0.125
+       15
$
$HMSET  1040400          10 "PLY1040400"
PLY     1040400 1          1.433486-45.0   YES      0.125
+       16
$$
$$  Stacking Information for Ply-Based Composite Definition
$$
STACK   1       SYM     1010100 1020100 1030100 1040100 1010200 1020200
+       1030200 1040200 1010300 1020300 1030300 1040300 1010400 1020400
        1030400 1040400
```





Design Variable Definition
With Manufacturing Constraints

```
DCOMP          1    STACK       1
+         PLYPCT      ALL
+        BALANCE     45.0   -45.0
ENDDATA
```

- DCOMP
  - Ply based sizing design variable definition
- Manufacturing Constraints are carried over from the Free Sizing Phase automatically with *OUTPUT,FSTOSZ,YES*

Output Request from Sizing Optimization

```
$$
$$ Optistruct Input Deck Generated by HyperMesh Version  : 9.0b121
$$ Generated using HyperMesh-Optistruct Template Version : 9.0b121
$$
$$   Template:  optistruct
$$
$$
$
OUTPUT,SZTOSH,YES
OUTPUT,PCOMP,YES
SCREEN OUT
$$------------------------------------------------------------------------------$
$$                       Case Control Cards                                      $
$$------------------------------------------------------------------------------$
$
$  OBJECTIVES Data
$
```

OUTPUT,SZTOSH,YES

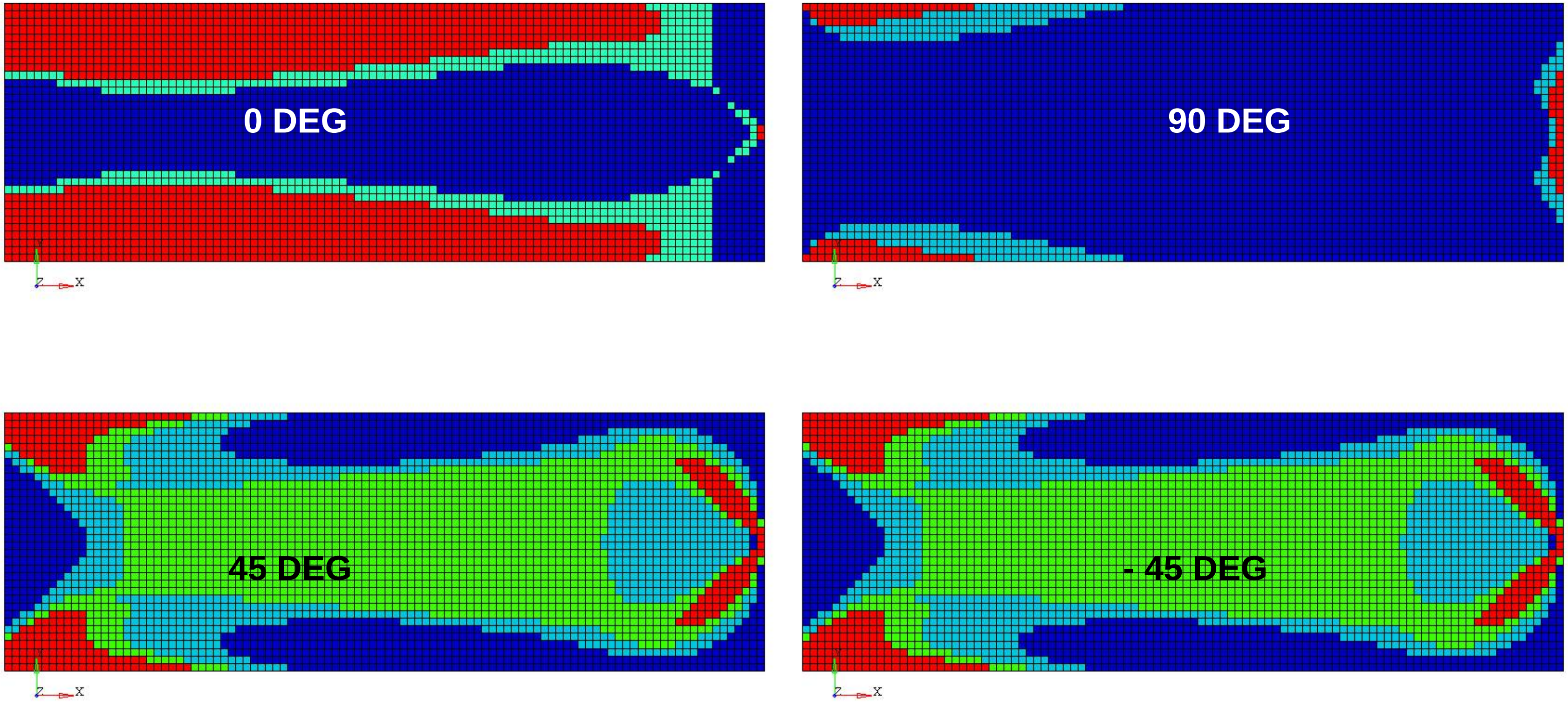
"Size to Shuffle"

Creates a new *.shuffling.fem deck

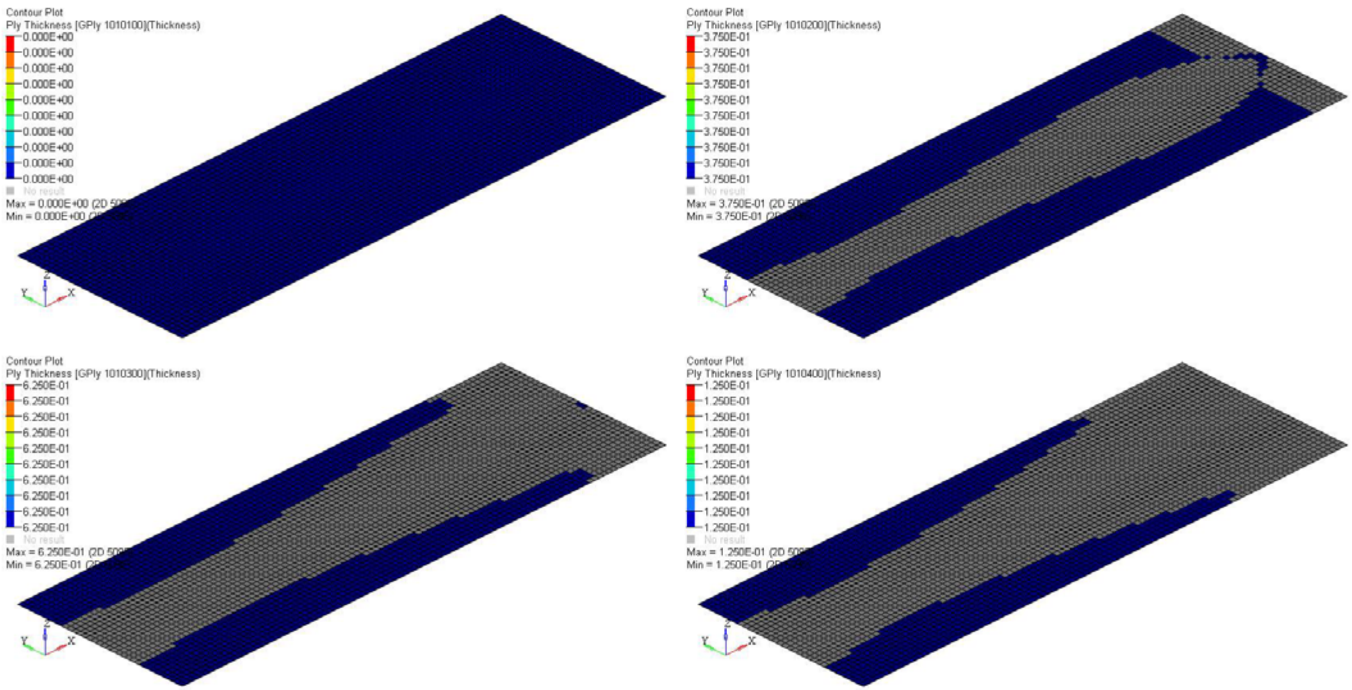
Activate output

Composite Optimization

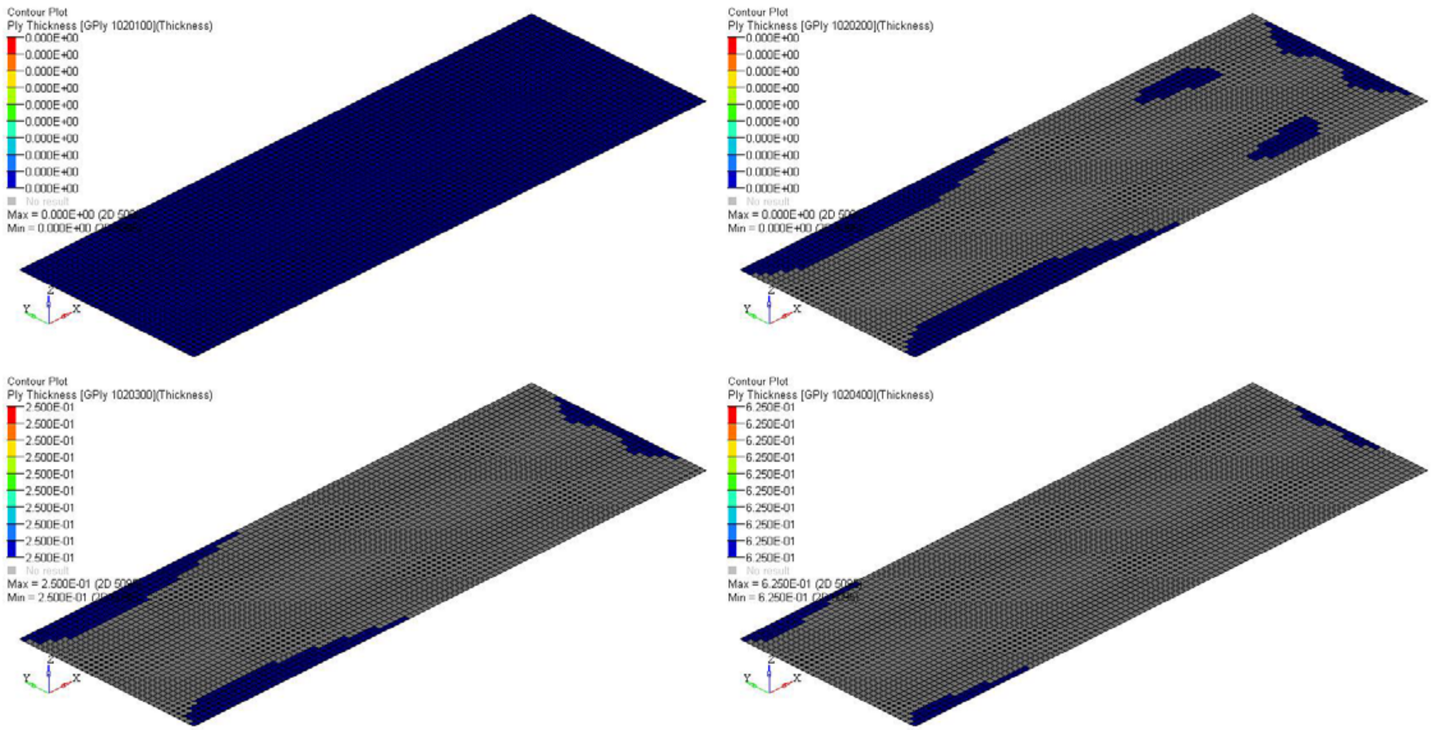
Size Optimization Results Per Fiber Orientation



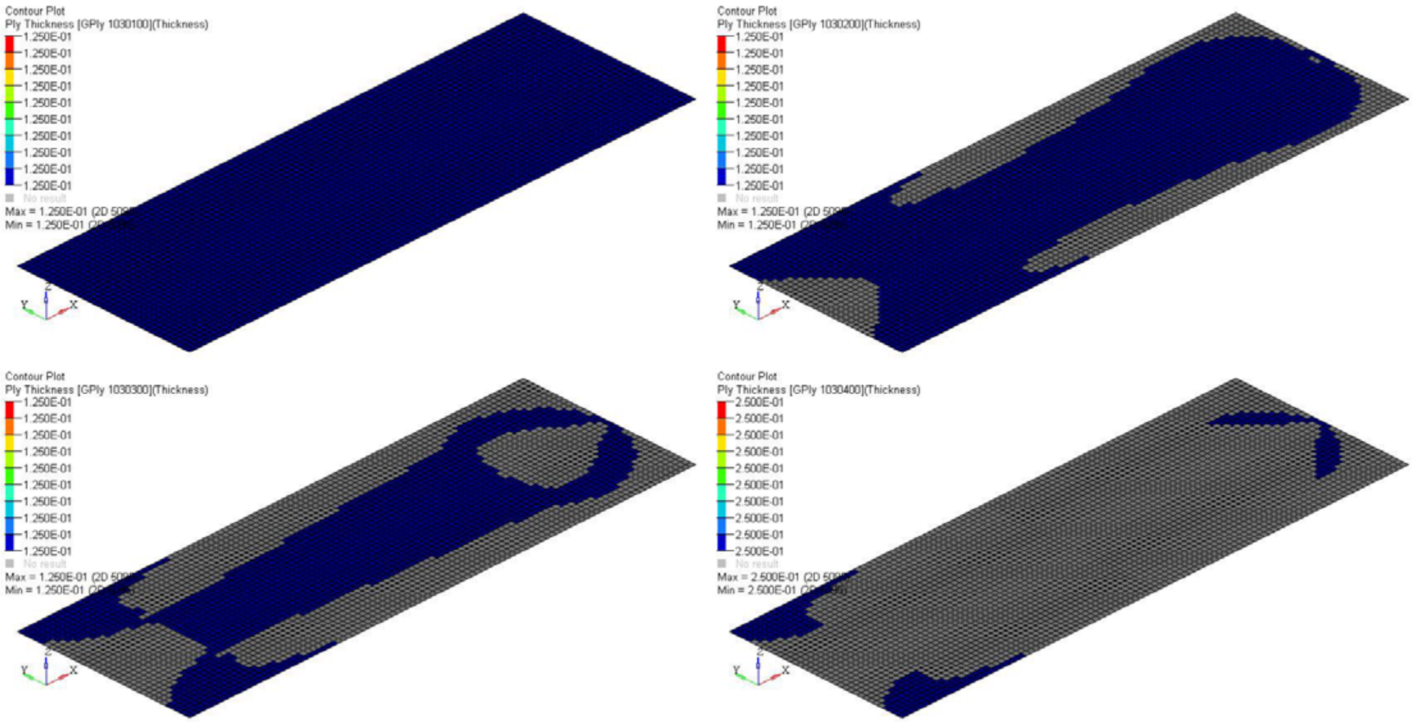
Optimized Ply Bundle Thicknesses: 0 Deg



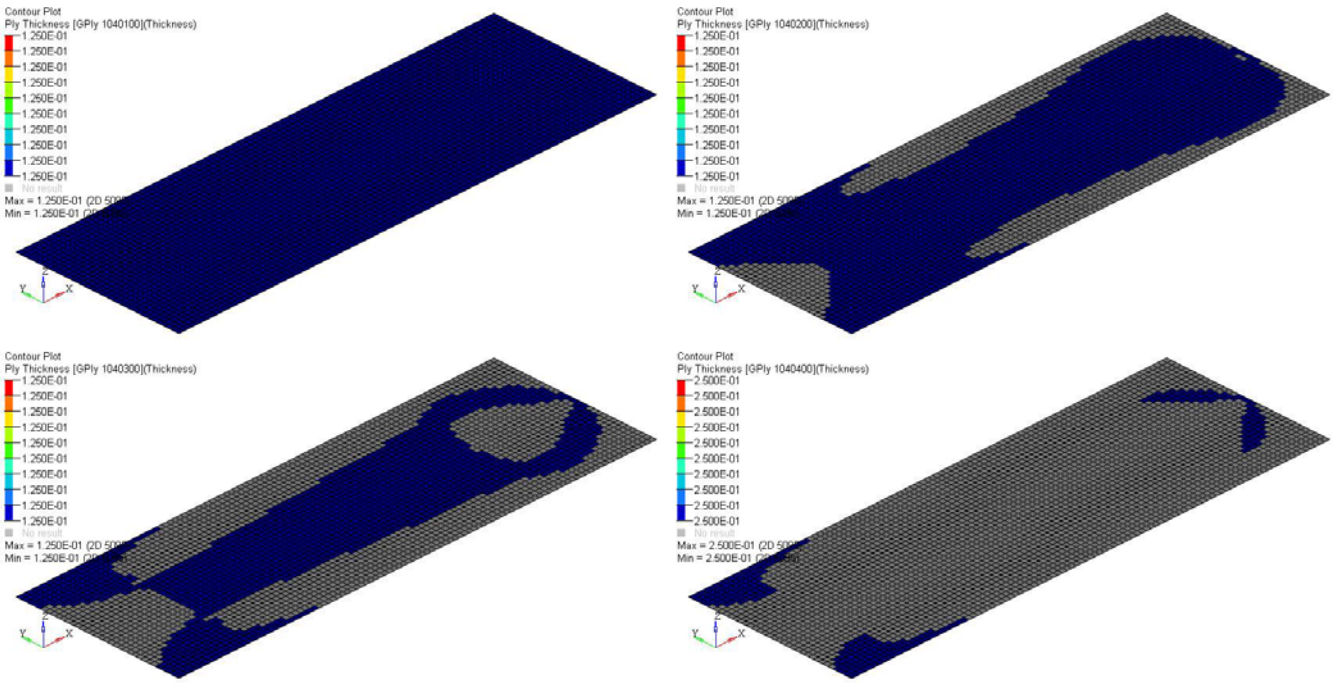
Optimized Ply Bundle Thicknesses: 90 Deg



Optimized Ply Bundle Thicknesses: +45 Deg

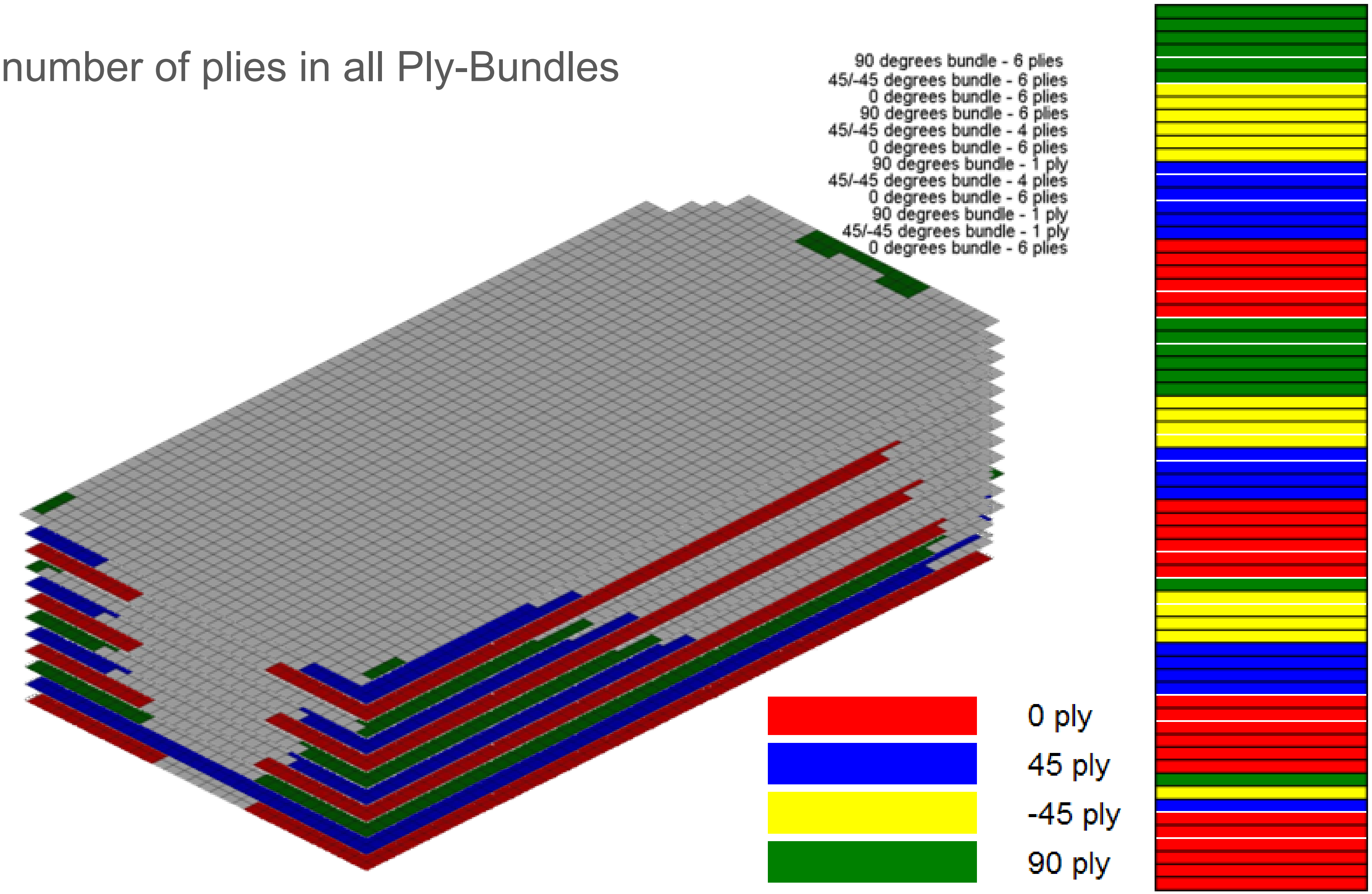


Optimized Ply Bundle Thicknesses: - 45 Deg



Phase II – System: Ply-Bundle Sizing

Final number of plies in all Ply-Bundles

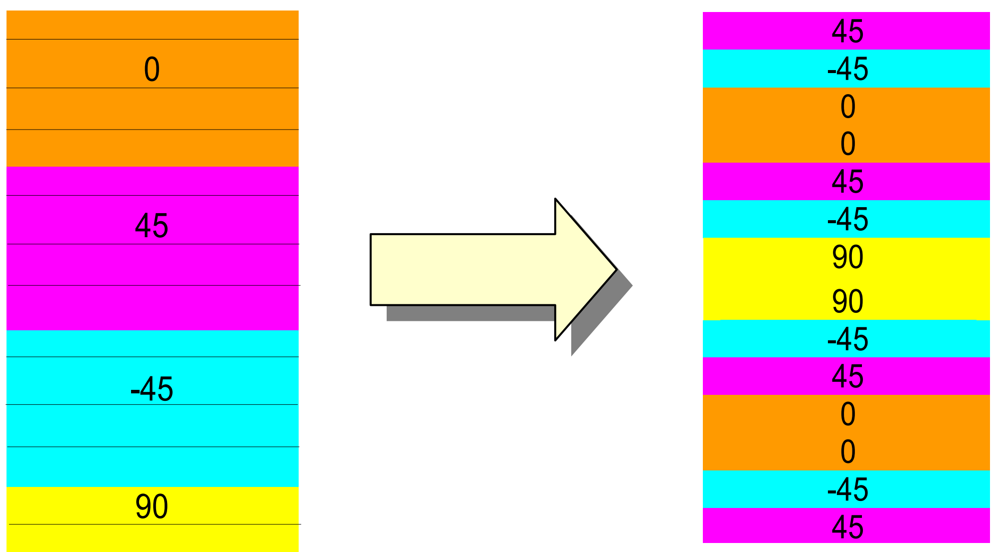


Composite Optimization

PHASE III

Detail: Stacking Sequence Optimization

- Meet ply book rules
- All behavior constraints
- Stacking manufacturing constraints



Ply Stacking Sequence Optimization

Cantilever Plate